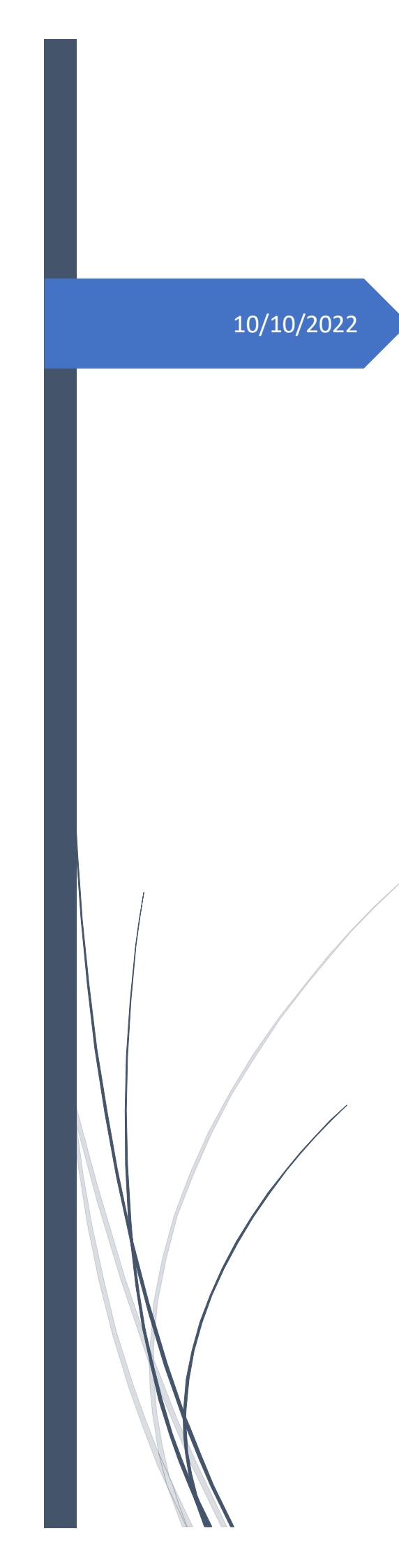




10/10/2022

OPTIMISATION OF A TANKER PROPELLER BLADE USING CARBON FIBRE REINFORCED PLASTIC

Structural & Fluid Analysis

Aron-Rhys Jones
2017757

Abstract

The purpose of this report is to optimise the propeller blades used on a tanker ship using composite materials. Whilst functioning the propeller will need must not exceed a maximum deformation of 100mm. This will be carried out using a finite element analysis software package known as Design Studio. This software allows for a variety of parameters that can be utilised to optimise the design of the blades effectively. The report will also contain a discussion on FEA techniques and how they are used to optimise designs. A prototype propeller will be analysed using the latest technology and techniques to gather relevant data towards the functionality of the blades.

Table of Contents

Abstract.....	1
Table of Figures.....	2
Introduction	3
Finite Element Analysis	4
Baseline Model Properties.....	6
Prototype Testing.....	8
Design Standards	9
Conclusion.....	9
Bibliography	10
Appendix	12

Table of Figures

Figure 1 Material Properties	6
Figure 2 Vector Orientation	6
Figure 3 Layer set up.....	7
Figure 4 Composite Blade	7

Introduction

The main focus of this paper is the use of Finite Element Analysis to produce an optimised design of a propeller blade to be used on a tanker ship. This design will be required to meet a maximum deformation of 100mm whilst using as little material as possible. The blades will be required to withstand harsh marine environments for long periods of time under constant stress. On average a tanker will travel for up to 20 days with the propeller almost running constantly (Rodrigue, 2020). These tankers on average use around 225 tonnes of fuel each day which results in a high cost per voyage. The tankers need to be as efficient as possible in order to reduce the consumption of fuel and to prevent delays in shipping times (Rodrigue, 2020).

As the original blade is made from steel it will be compared to the new carbon fibre reinforced plastic (CFRP) blade to determine if it has a reduced weight. The initial CFRP used will be at a thickness of 0.1mm per ply. A ply refers to the thickness of a CFRP layer. These layers are bonded together in a furnace which cures the epoxy to the carbon fibre in order to produce a light weight and exceptionally strong material (F.Al-Mahmoud, 2018).

The design of the blades will be optimised using Design Studio which is a CAE software. This will allow for the material thickness to be calculated at each orientation of the propeller blade. The report will also be evaluating the different methods of FEA optimisation available and discuss how it has developed. The future development of FEA will be discussed and how it can impact current and future industries. This paper will discuss prototype testing for the propeller blade and the recommendations on how the testing should be conducted and how the data should be acquired.

The British standards that surround the designing and testing phases will be considered to ensure that the blade is in compliant and able for use within British industries.

Finite Element Analysis

Origins of FEA

The usage of Finite Element Analysis (FEA) has become gradually more significant as technology has improved over the last century. The incorporation of FEA into FEM (Finite Element Method) was first implemented in the 1940's. This allowed for IBM and General Motors to create a computer system called DAC-1 to facilitate the designs of cars (Crahmaliuc, 2020) (Moaveni, 1999).

The creation of this system guided NASA to the development of their own structural analysis solver tool named NASTRAN which became known as the first generation of FEA software (Actis, 2018). NASTRAN was selected to replace the existing analysis tools that were used for structures and composite aircrafts. NASA was then able to implement standards for modelling and simulations which are still followed in the current industry codes of practice (Crahmaliuc, 2020) (Tipton, 2017).

Originally, a concept design would need to be hand drafted which would take a substantial amount of time. This required suitable tooling to be manufactured in order for the design to be manufactured (Dyi-Cheng Chen, 2018). Testing could only be carried out after the manufacturing of the design. The means that any errors with the design would result in the repeating of the process until an optimised design was achieved (A.Gnanavelbabu, 2014).

Advantages of FEA

Finite Element Analysis has an increased accuracy over the traditional methods of using physical prototypes and stress testing. The process allows for a designer to input the required part and model the physical stresses that will affect it. This will also show any potential stresses that may have been overlooked during the initial design process thus improving the accuracy of the design. The software has the ability to change the material of the model with ease to show how an alternate material could improve or worsen the design (Bi, 2019). This removes the need for multiple prototypes of varying materials being manufactured. The reduction in the number of prototypes required will save time and money for the company. The testing of models can be carried out within a matter of hours using FEA as opposed to days or weeks using physical prototypes and manual calculations.

Future of FEA

The potential of Finite Element Analysis is that it will be able to be used within the additive manufacturing markets and that it will also be able to utilise multi-materials. The optimisation of a 3D concept could be reduced depending on the size to as little as a day (Wood, 1992). Future developments will lead to the solvers becoming more reliable and will remove the need for any physical testing (Bhaumik, 2016). The current limitations of FEA are the fluid-thermal topology, multi-material topology and fluid-structure topology. Once these are solved it can be combined with additive manufacturing leading to the next generation of industry and using artificial intelligence is a near prospect (Roade, 2020).

FEA Tests

There are various tests that can be carried out during finite element analysis. The first type is structural static analysis which analyses scaled models based on dimensions to determine that a small-scale structure will be able to withstand the same interactions as a full-scale structure (Brush, 2022). Another test is Thermal engineering analysis which investigates different variations in temperature to determine the effects on the structure of the design. Modal analysis is used to examine the effects caused by external disruptive vibrations on the design's structure (Baqersad,

2014). Finally, engineering seismic calculations are used to test the performance of structures that are subjected to an array of ground frequencies to ensure that the final design structure is capable of withstand them (Brush, 2022).

FEA Techniques

The technique that will be used for the propeller blade will be topology optimisation as found within Design Studio. This method optimises the material layout and how it is distributed in a given area. The given area is then analysed for any stresses that are present and then displays the different section that are being exposed to stress and strain (Krist V. Gernaeya, 2015).

Baseline Model Properties

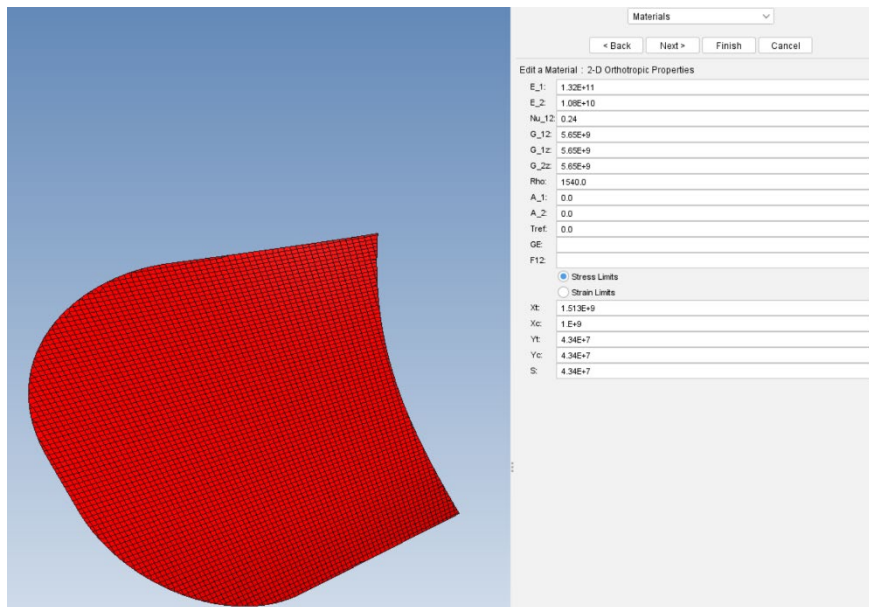


Figure 1 Material Properties

The material properties of the blade required changing from steel to carbon fibre reinforced plastic (CFRP). This was achieved within the materials section and specifying a new material. CFRP is a composite material which is defined as 'MAT8/PSHELL/PCOMP' within Design Studio. This identifies the algorithm that is used for orthotropic materials within the software.

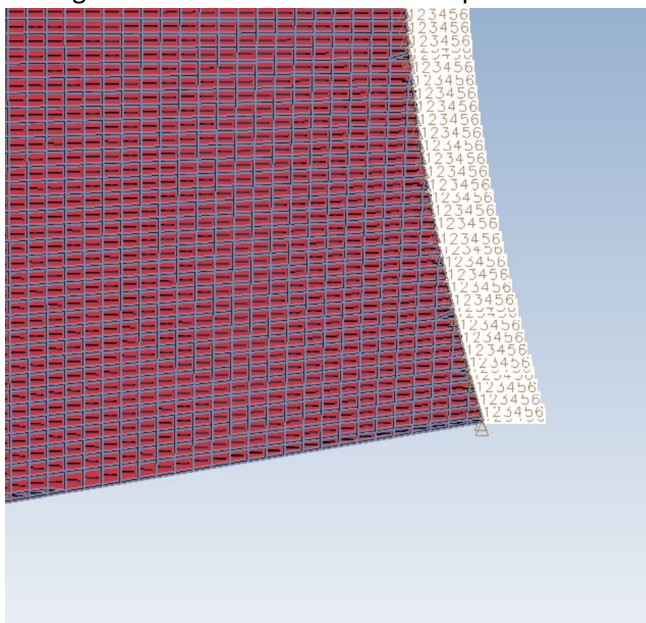


Figure 2 Vector Orientation

In order to guarantee a high level of accuracy the element properties of the model were set to the proper orientation. This was achieved by entering the 'elements' section and changing the elements theta to that of the Z plane.

Edit a Group : Choose Composite Property Values

Name:

Z0:

Nonstructural Mass:

Tref:

GE:

Interlaminar Shear Limit:

Failure Theory:

Laminate:

Type:

Global Ply ID	Material	Thickness	Angle	Output
Layer 1				
857142 UD	5.E-4	-90.0	Yes	+ -
Layer 2				
857142 UD	5.E-4	-45.0	Yes	+ -
Layer 3				
857142 UD	5.E-4	0.0	Yes	+ -
Layer 4				
857142 UD	5.E-4	45.0	Yes	+ -
Layer 5				
857142 UD	5.E-4	90.0	Yes	+ -

Figure 3 Layer set up

The next step is to set the thickness and the angles of the composite material. The material is set to the angles of -90° , -45° , 0° , 45° and 90° which will allow for the thickness of the material to be consistent all through the design model. The material thickness will be a laminate of 25 ply's of CFRP in which 1 ply is 0.1mm.

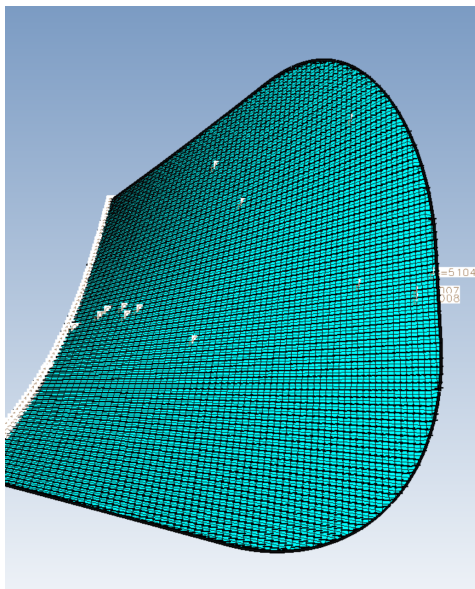


Figure 4 Composite Blade

Prototype Testing

The objective of creating a physical prototype is to demonstrate that the blade functions as indeed when compared to the simulated version. Before to any prototype testing health and safety precautions and regulations must be carried out. As the physical scale of the blade may be rather large lifting regulations (LOLER) will need to be followed to make sure handling and transportation are safe (HSE, 2013). The correct PPE must be worn when interacting with the blade as there may be a sharp edge. The testing area will require suitable guarding to prevent access during operation. The tests carried out will also need to be conducted in an environmentally controlled area to ensure that each test is consistent.

The test should be carried out on both a prototype steel propeller and prototype CFRP propeller to have a comparison between both sets of simulations. The propellers will need to be exposed to a variety of water temperatures to simulate the different marine environments they'll be used in. The propellers will be required to operate at the required RPMs for long periods of time. The prototypes will need to be exposed to higher RPMs than they will be used at to find their point of failure.

To test the deflection of the blades both prototypes will need to be subjected to high velocities of water. This will expose them to high levels of strain and stress to find the failure points. If there are any deflections or deformations during the testing the water flow will change. This can be monitored using a flow meter.

There are two forms of testing that can be used on the propeller blades these being pneumatically and hydraulically. The blades could be placed within a wind tunnel and compressed air can be used to simulate water flow.

Due to the advancements of data capturing the prototypes can be monitored and measured using a variety of sensors and equipment. The automation of data gathering has allowed for vast amounts of data to be gathered within short periods of time. By using manual instrumentation to capture data a large quantity of data is lost between gatherings as it is a lengthy process (Malak, 2022). Another issue with manual gathering is that there is the potential for human error which will affect the results.

The data gathered by the use of automated sensors and equipment can then be compared to the results of the analysis that has been carried out on Design Studio. This will show if there are differences between the prototypes and the simulated models.

Design Standards

Legislation

Due to Brexit any manufactured component produced after the 1st of January 2021 will need to be issued a UK Conformity Assessed marking (UKCA) which has replaced the previous CE marking (Gov, 2020).

Depending on the final scale of the propeller the design will have to comply with either ISO 484-1:2015 or ISO 484-2:2015. These are the manufacturing tolerances for ship propellers of diameters greater than 2.50m and for propellers between 0.80m and 2.50m (ISO, 2015).

Conclusion

From the information that has been gathered in this report it shows that Finite Element Analysis is important to current and future industries. FEA has been proven to be more accurate than other methods of designing models due to its advanced ability to show precise areas of strain on a part. It has greatly reduced the time that it takes to complete the design process and has cut down the costs of labour and materials for the process. Prototypes are crucial to the process even with FEA as it allows for the comparison between the simulated data and the physical data. The prototype testing should be carried out using automated data gathering equipment to limit the number of errors that can occur and to compile a large amount of data. The live stream of data can allow for the exact points of failure to be obtained which cannot be done using manual gathering methods. The design of the blade will need to comply with the current British standards regarding ship propellers and how they are manufactured.

Bibliography

A.Gnanavelbabu, P., 2014. Identification of Major Lean Production Waste in Automobile Industries using Weighted Average Method. *Procedia Engineering*, 97(1), pp. 2167-2175.

Actis, D. R., 2018. *Finite Element Modeling vs Numerical Simulation*. [Online]
Available at: <https://www.esrd.com/simulation-technology/brief-history-of-fea/>
[Accessed 15 October 2021].

Baqersad, J., 2014. Comparison of Modal Parameters Extracted Using MIMO, SIMO, and Impact Hammer Tests on a Three-Bladed Wind Turbine. In: *Topics in Modal Analysis II*. s.l.:s.n., pp. 185-197.

Bhaumik, D., 2016. *How Finite Element Analysis is Helping Design the Future*. [Online]
Available at: <https://www.hitechfea.com/fea-knowledgebase/how-finite-element-analysis-is-helping-design-the-future.html>
[Accessed 20 10 2022].

Bi, Z., 2019. *Finite Element Analysis Applications*. 1st ed. Indiana: Academic Press.

Brush, K., 2022. *TechTarget*. [Online]
Available at: <https://www.techtarget.com/searchsoftwarequality/definition/finite-element-analysis-FEA#:~:text=Various%20different%20types%20of%20tests%20are%20used%20during,Modal%20analysis.%20...%204%20Engineering%20seismic%20calculations.%20>
[Accessed 20 10 2022].

Crahmaliuc, R., 2020. *75 Years of the Finite Element Method (FEM)*. [Online]
Available at: <https://www.simscale.com/blog/2015/11/75-years-of-the-finite-element-method-fem/>
[Accessed 20 10 2022].

Dyi-Cheng Chen, L. C.-Y. a. Y.-Y. L., 2018. Finite element analysis of deep drawing. *Advances in Intelligent Mechatronic Systems and Precision Engineering*, 11(9), pp. 1-10.

F.Al-Mahmoud, 2018. 3.22 CFRP Reinforcement Rods. *Comprehensive Composite Materials II*, 3(1), pp. 578-591.

Gov, 2020. *Using the UKCA marking*, London: Department for Business, Energy & Industrial Strategy.

HSE, 2013. *Lifting equipment at work*. [Online]
Available at: <https://www.hse.gov.uk/pubns/indg290.pdf>
[Accessed 20 10 2022].

ISO, 2015. *Marine engines and propulsion systems*. [Online]
Available at: <https://www.iso.org/ics/47.020.20/x/>
[Accessed 20 10 2020].

Krist V. Gernaeya, J. M. W. U. K., 2015. Topology optimization for biocatalytic microreactor configurations. *Computer Aided Chemical Engineering*, 37(2), pp. 1463-1648.

Malak, H. A., 2022. *What is data capture and why is it important?*. [Online]
Available at: <http://theecmconsultant.com/what-is-data-capture/>
[Accessed 20 10 2022].

Moaveni, S., 1999. *Finite Element Analysis: Theory and Application with ANSYS*. 5th ed. s.l.:Prentice Hall.

Pagadala, P., 2015. *Topography Optimization*, France: Altair.

Roade, G., 2020. *THE FUTURE OF FINITE ELEMENT ANALYSIS: MESHFREE*. [Online]
Available at: <https://www.midasoft.com/mechanical/blog/the-future-of-finite-element-analysis-meshfree>
[Accessed 20 10 2022].

Rodrigue, J.-P., 2020. *The Geography of Transport Systems*. Fifth ed. New York: Routledge.

Tipton, R., 2017. A Brief History of Finite Element Analysis. *Design Engineering, Digital Simulation, Engineering*, 1(1), p. 1.

Velling, A., 2020. *Topology Optimisation*. [Online]
Available at: <https://fractory.com/topology-optimisation/>
[Accessed 20 October 2021].

Wood, J., 1992. The Future of Finite Element Analysis in Undergraduate Mechanical Engineering Courses. *Engineering Simulation Education & Competence Framework R&D*, 1(1), pp. 1-10.

Appendix

Appendix A – Material properties set up of CFRP

Materials

< Back Next > Finish Cancel

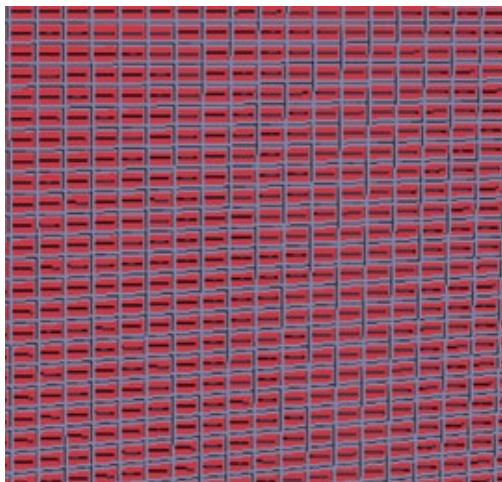
Edit a Material : 2-D Orthotropic Properties

E ₁ :	1.32E+11
E ₂ :	1.08E+10
ν ₁₂ :	0.24
G ₁₂ :	5.65E+9
G ₁₃ :	5.65E+9
G ₂₃ :	5.65E+9
ρ ₀ :	1540.0
A ₁ :	0.0
A ₂ :	0.0
Tref:	0.0
GE:	
F12:	

☒ Stress Limits
☐ Strain Limits

Xt:	1.513E+9
Xc:	1.E+9
Yt:	4.34E+7
Yc:	4.34E+7
S:	4.34E+7

Appendix B – Z Vectors



Appendix C – Composite angles

Edit a Group : Choose Composite Property Values

Name: Laminar

Z0:

Nonstructural Mass: 0.0

Tref:

GE:

Interlaminar Shear Limit:

Failure Theory: None

Laminate: All Plies

Type: Regular

Global Ply ID	Material	Thickness	Angle	Output
Layer 1				
857142 UD	5.E-4	-90.0	Yes	+ -
Layer 2				
857142 UD	5.E-4	-45.0	Yes	+ -
Layer 3				
857142 UD	5.E-4	0.0	Yes	+ -
Layer 4				
857142 UD	5.E-4	45.0	Yes	+ -
Layer 5				
857142 UD	5.E-4	90.0	Yes	+ -