

2022

Structural and Fluid analysis

FINITE ELEMENT ANALYSIS
DANIEL THOMAS

Contents

Introduction	2
FEA optimization techniques	3
A Brief History	3
Future of FEA optimisation and impact on the design process	4
Using FEA optimisation on a ship's propeller	6
Laboratory testing of a prototype	11
Technical literature and standards of a propeller	12
Conclusion.....	13

Introduction

This report demonstrates how finite element analysis optimisation or FEA optimisation has developed over recent years and how it can be used today to aid product design. FEA optimisation has been used in the design of a ship propeller by using carbon fibre reinforced plastic or CFRP. This report compares the use of steel and CFRP for this purpose and the effect that optimisation has in increasing the suitability of CFRP for this purpose.

Finally, discussion is given at how a blade could be tested, data collected and then compared with the simulation from the FEA optimisation. And then a discussion is given for any technical literature and standards that would be related to the ship's propeller.

FEA optimization techniques

There are many different techniques available to engineers to optimize a component or assembly. These techniques have improved over time to make the process more efficient, by saving time and ultimately money.

The techniques have changed over the years, going from hand calculations to full FEA optimisation software. Further on, the history of FEA is discussed along with current and future techniques.

A Brief History

Finite element analysis has developed over the year due to more processing power from computers and the needs of the aerospace industry.

Finite element analysis, FEA, began initially around the 1940s by Richard Courant and Hreinkoff. It was developed to solve structural analysis problems within the civil and aeronautical industry, **Radu Crahmaliuc (2020)**. This method involved breaking down the system in question into smaller triangular sections to carryout calculations on them. This allowed them to solve very large problems efficiently and accurately.

With the development of commercial computers in the 1950s, FEM developed to solve much more complicated structural analysis on frame and truss systems. A paper was also written during this period which described the concept of Finite element analysis, and thus 'finite element' was coined first. Also, during this period, Boeing were heavily investing in FEA and computer power, and developed their own programs to analyse the stress in aircraft wings using triangular stress elements.

During the 1960s each industry and company had their own codes for finite element analysis. NASA had awarded a contract to a small computer manufacturer to create a general-purpose FEA code that could be used nationally. The company developing the code developed NASTRAN and after the contract ended the company went forward to develop MSC/NASTRAN. NASTRAN had then become available to the public, **Radu Crahmaliuc (2020)**.

During the 1970s the FEA solvers had a limit of 68000 degrees of freedom, and not much develop further due to the lack of performance from computers, **O'Donnell Consultancy (2018)**.

In the 1980s, graphic improvement on computer meant that FEA solvers could develop to include colour coding. This gave engineers a graphical display of how stress/strain would affect components without needing to analyse raw data. Again, with the development of the computer, the capabilities of FEA solvers have got better, allowing for more calculations to be solved at a quicker rate.

Since the 1980s, FEA solvers have become much more powerful, in the 1990s, allowing models to have 1 million degrees of freedom, whilst running the solver on an ordinary computer.

Future of FEA optimisation and impact on the design process

FEA solvers continue to develop today. It is now possible to run solvers on cloud-based solutions, which would not require powerful desktop computers, and allows engineers to run models in the background whilst carrying out other work. In future it may be possible for FEA solvers to calculate environmental impact of assemblies due to certain materials that are used or amounts of materials as companies are being pushed to calculate the carbon footprint of their products during manufacturing through to disposal.

FEA solvers have allowed engineers to become much more efficient, cutting down time to solve problems and cut down times when developing components and assemblies. Historically, components would be designed through experience and certain size/geometrical constraints, manufactured and then tested. If it failed the tests, it would be redesigned, remade and retested. This cycle would go on until the component or assembly would pass the tests.

This method could be quite time consuming, and costly not just in labour but also materials to produce the component to test. This also has an environmental impact due to the waste during the development of parts.

Not so long ago, with the introduction of commercial FEA solvers, the process to design parts would be led by CAD. The component would again be designed within CAD through experience, then tested in FEA. This would feedback to the design back to CAD. Once the engineer was confident with the design, the component would be manufactured and then tested, typically it would pass the test. This vastly reduced the length of time taken to design a component, reducing costs in tooling and components and had a positive impact on the environment.

Nowadays, components are designed within the FEA solver and tested within the solver. Once the engineer is happy with the design, it is then transferred into CAD to produce drawings etc. Then components are produced and tested. Again, this cuts down the design process time, cutting costs and cutting the environmental impact.

Currently there is a push for companies and individuals to cut their carbon footprint as much as possible and with rising manufacturing costs, and shortage of certain materials/components, it is now more important than ever to consider using software packages such as this to design components. Using FEA solvers allows engineers to design and verify their designs before any material is needed to test.

For large assemblies such as ships, buildings, bridges etc, it is essential to use this to understand if the design would work. Before FEA solvers, engineers would have calculated stress/strain for these assemblies, and built models to understand how they would work, but models are limited. Models may not behave in the same way as the full-size version would, better results will be had with an FEA solver.

The optimisation within the FEA solvers will change the topology, topography, composite, size and shape of the design to achieve the given goals, and objectives can be set to reduce weight or material usage. This can trim down over engineers' components and assemblies whilst retaining the key objective.

There is downside to the FEA solvers/optimisation. Firstly, to use these packages effectively, experience is required, as its easy to make mistakes that the software won't pick up. It could cause a badly designed part to be manufactured, which would eliminate the time gained through using FEA.

Also, the initial cost of implementing FEA optimisation into an organisation can be high, cost of equipment, software, training and an experienced FEA engineer. This cost will however be recovered in efficiency in the design process.

Using FEA optimisation on a ship's propeller

When designing the lay of the CFRP propeller, it is important to get a baseline. This will be set by carrying out an analysis on a steel propeller with the same dimensions. The analysis will be carried out on a single propeller.

The load on the propeller has been set to 2424 newtons as a UDL across the surface of the model, in the Z direction. This force has been decided as from research seemed to be a similar amount of force exerted on ship propellers, and through the FEA results were comparable to research. As is written in the **HydroComp (unknown date)**, states. These propellers undergo 3 types of forces, thrust, centrifugal and torque. It was decided not to add this as individual force on the blade during the FEA optimization as the exact figures are not known.

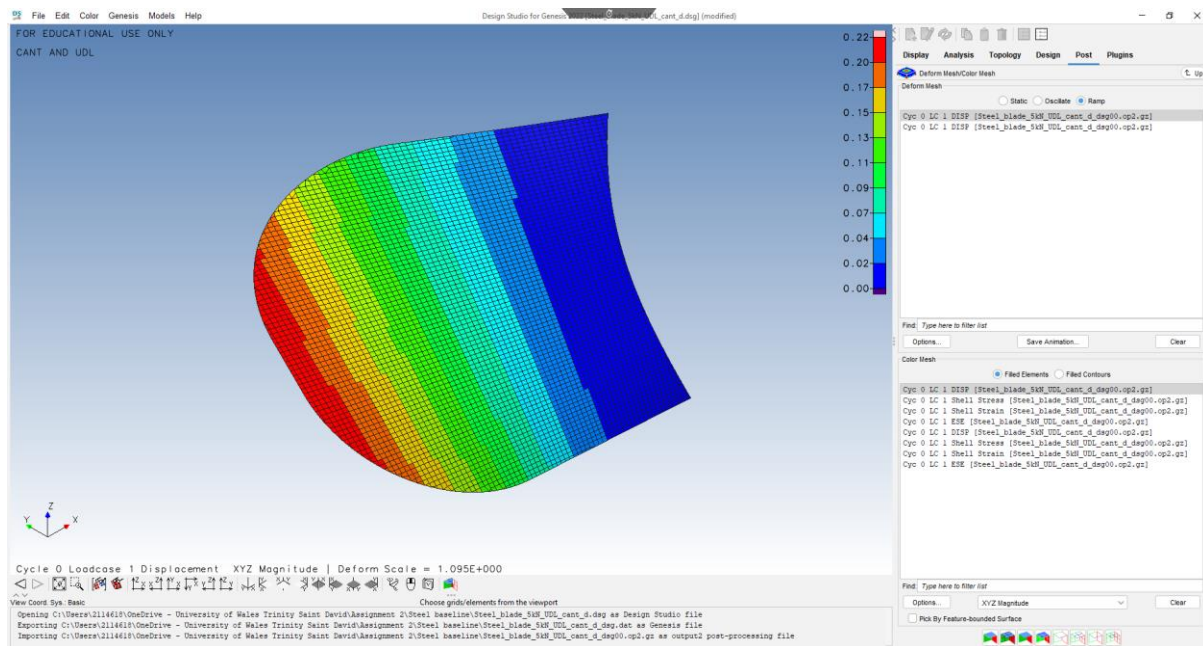


Figure 1

Above, in figure 1, shows the deflection of the blade. The maximum deflection on the blade is 220mm at the tip of the blade. This is what would be expected as the tip is the further away from the fixed hub. The total weight of this blade in steel is 646kgs. From this information, the stiffness and stiffness to weight ratio can be calculated. These results are tabled below figure 8.

The next step of the design process is to change the material to CFRP, using standard material properties.

The CFRP is to be layered, and each layer is in a different orientation. The orientations are as follows, 0 degrees, in the Y direction, 45°, 90°, -45° and -90°. Each layer is 0.1mm thick and initially there will be 5 layers in each direction.

Below shows the single analysis results, figure 2.

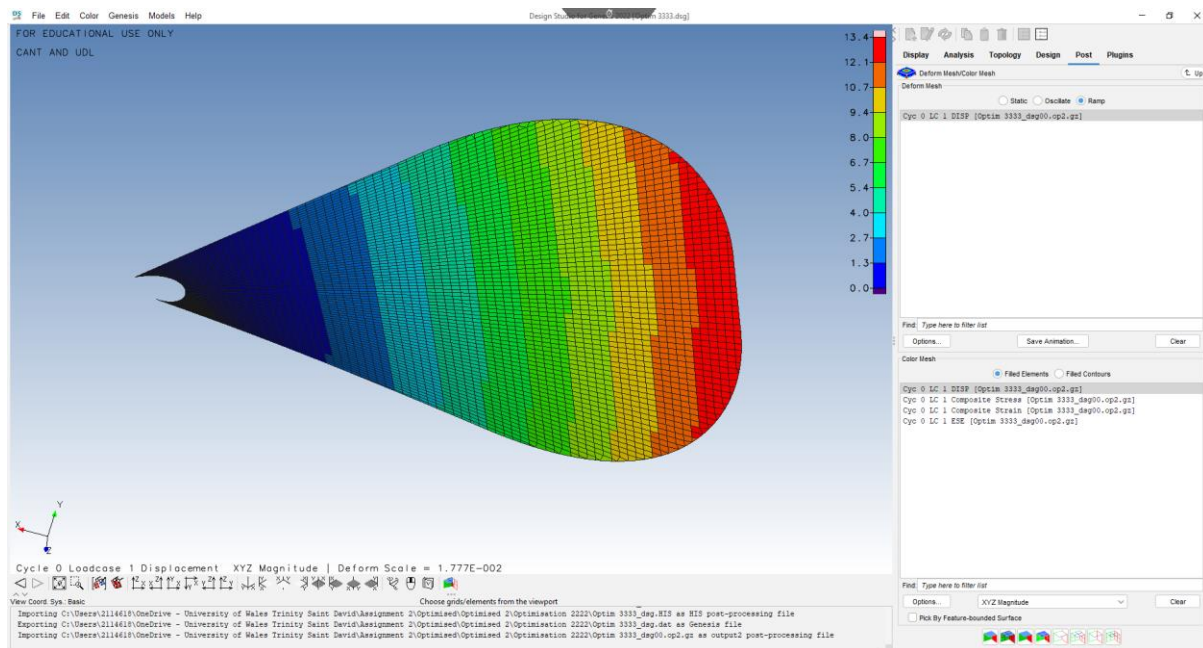


Figure 2

The total thickness of this blade is 2.5mm and weighs 32 kgs. Looking at the deflection, again maximum deflection occurs at the tip of the blade. This blade deflects 13 meters. Clearly using this type of lay for the CFRP blade would not work as a propeller.

The next stage of the design process is to optimize the lay of the CFRP. The objective was to achieve a maximum deflection of 100mm whilst keeping the mass of the propeller to a minimum. Also, within the software the design variable of the layers were also set up. Below it was set up to have a maximum of 15 layers in each direction, with at least 1 layer in each direction.

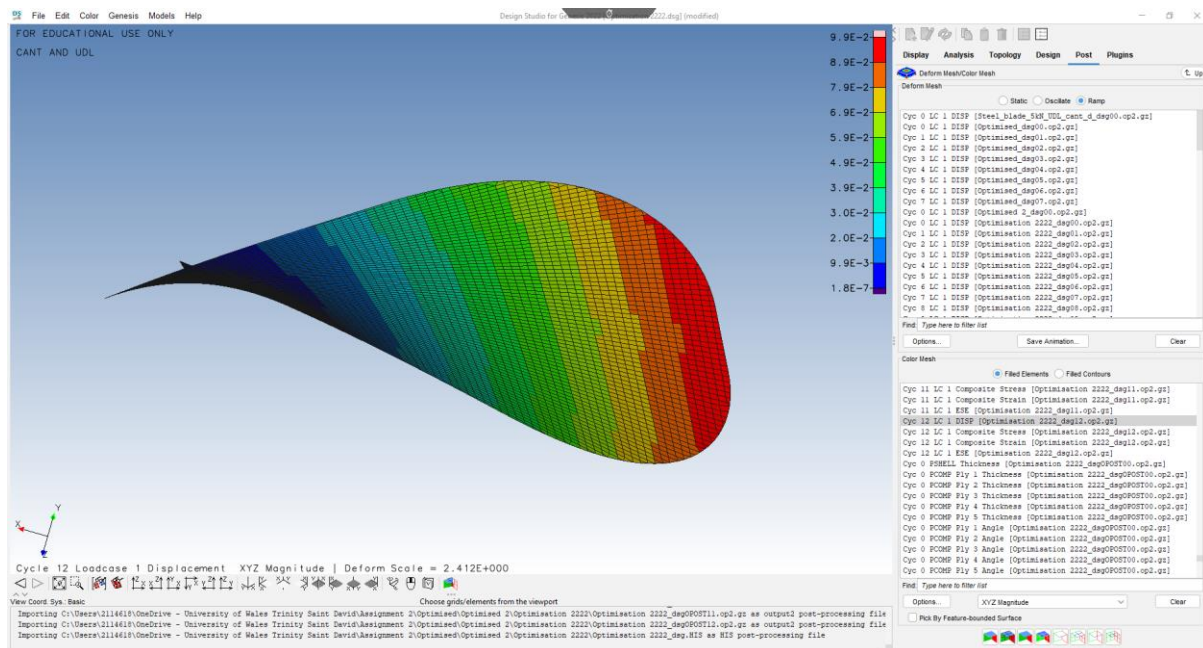


Figure 3

Above in figure 3, shows the deflection of the propeller. Again, maximum deflection is achieved at the tip of the blade, but this time the deflection is within the specified limits at 99mm, and the mass of the propeller is 164kgs. The graphs below show how the software has changed the number of layers in each direction to get within the objectives.

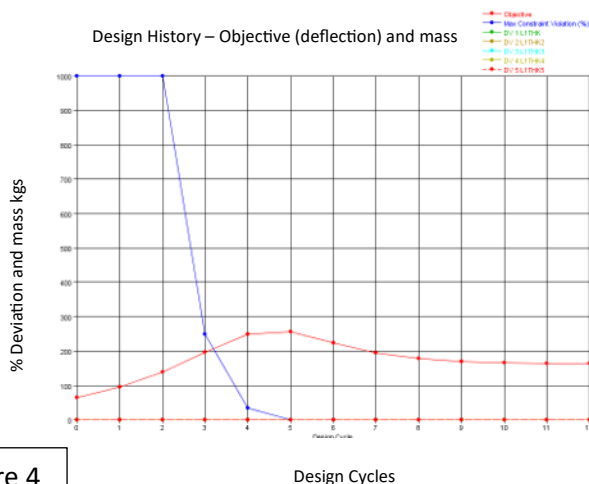


Figure 4

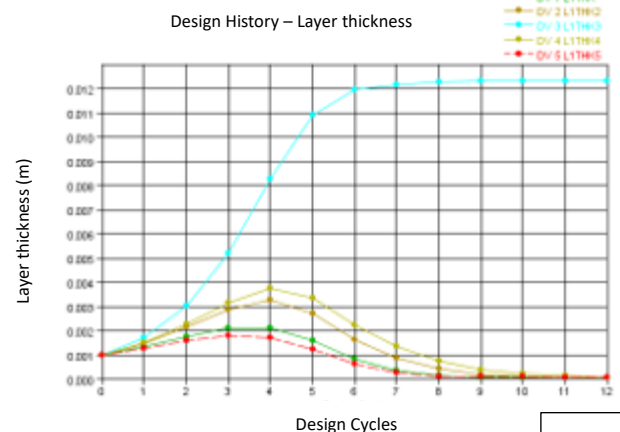


Figure 5

The graph to the right, figure 5 shows how the software was adding thickness to each direction and then tests the deflection and mass. The X axis is the number of design cycles, and the Y axis is layer thickness in meters. The graph shows that the software has achieved the objectives within 12 cycles. The graph shows that the software has added most layers in the 0° direction as it has the most impact then trying to reduce deflection. The software has also removed layers from the other directions to keep the mass at a minimum whilst keeping the deflection to under 100mm.

The graph on the left, figure 4, shows the objective violation (blue) and mass in kilograms (red). Again, the X axis is design cycles, and the Y axis is a value for percentage deviation and mass in kg. Comparing both graphs together, it can be seen the objective violation falls when layers are being increased in the 0° direction. It can also be seen that the mass increases when adding layers in all the

other directions, which the software then reduces as they do not affect the deflection as much. The weight then reduces with less layers in 45° and 90°.

The objectives have been met with this optimization; however, these layers are not manufacturable. To be manufacturable, the layers are 0.1mm thick, the software has not been set up to increase the layer by 0.1mm thick, so some layer may be 0.14556mm, which would be impossible to manufacture. In the next analysis the software has been set up to increase or reduce layers by 0.1mm. Below are the results.

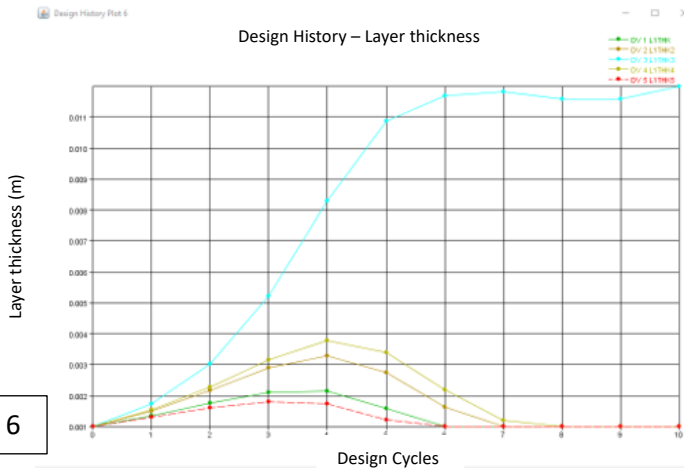


Figure 6

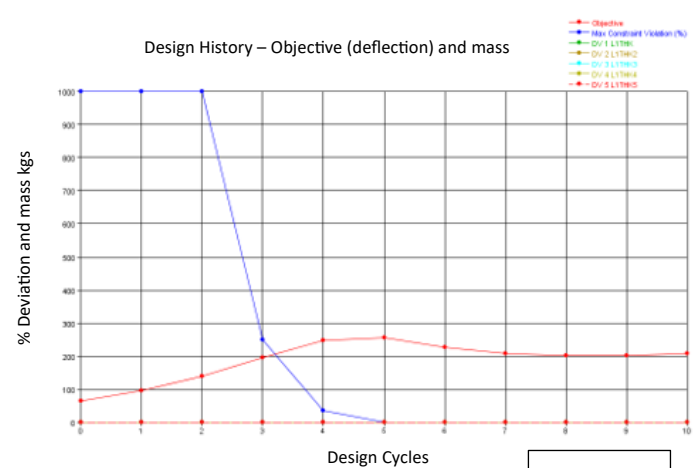


Figure 7

The graphs above look like the previous analysis. The deflection of the blade is now 90mm and the weight has increased to 206kgs. This is due to limiting the software to +/-0.1mm layers. Again, the software has added most of the layers in the 0° direction and has reduced the layer thickness in other directions to control the mass of the propeller. The optimization of the CFRP was achieved in 10 cycles.

The table below shows the deflection, stiffness and stiffness to weight ratio of each simulation.

	Deflection (mm)	Mass	Stiffness	Stiffness to weight ratio
Steel baseline	220	646	11.01	0.0017
CFRP 5 layers each direction	1340	32	1.81	0.0058
CFRP optimization	99	164	24.48	0.1522
CFRP optimization final	90	206	26.93	0.1333

Figure 8

Using this table above it can be easy to compare the different materials, and the different lay of the CFRP. A greater stiffness to weight ratio is important for many reasons. As can be seen, CFRP outperforms steel in stiffness to weight ratio, even the unoptimized CFRP. However, we know that this CFRP design would fail as it deflects nearly 13 meters. The ideal solution is the initial optimization, as it has a stiffness to weight ratio of 0.1522 N²/mm, however as described previously, this type of design is not manufacturable

Through the optimization techniques used on Design Studio, CFRP looks to be a viable alternative to steel, and would have many advantages. Due to CFRP being the 3rd of the weight of steel, less power would be required to drive the propeller, it could possibly mean less force exerted on the drive shaft and bearings. This would mean a more fuel-efficient ship, that would require less maintenance, saving money.

Laboratory testing of a prototype

Testing of the prototype should be carried out on a full-scale blade, as when testing for deflection, a scale model will not be accurate. This would be for many reasons such as the fibres of the materials not being scaled down, the effect of gravity on a scale model would be different, and of course the mass of the scaled down model would not be the same.

When the testing is carried out, the blade should be held rigidly at the edge that would be fixed to the hub of the propeller. This will need to be secure as to not deflect when load is applied to the blade, as this will not give an accurate value for deflection.

In the simulation, the only force that was applied to the blade was a UDL across the blade at a total value of 2424 Newton meters. As was discussed earlier in the report, there are three main forces at work on the blade during use, these are gravitational, torque and centrifugal.

Initially the testing should be carried out to replicate the results seen in the simulation. The blade should be constructed from CFRP, and the layers built up the same as seen in the last simulation as accurately as possible. If layers are not built up the same orientation or quantities, then results will differ.

A load will need to be applied on the blade in the same direction as the simulation. A load of 2424 Newtons will need to be applied in the calculated position to have the same effect as a UDL across the blade. This load can be calculated using the cantilever beam deflection.

A datum should be set up at the tip of the blade that is independent from the blade before the load is applied, and once applied, the deflection can be measured from this point. From here the deflection can be compared with the results gained from the simulation.

Strain gauges could also be placed down the length of the blade at key points, using the simulation as a guide. As can be seen from the simulation, very little deflection was seen at the fixed end of the blade, and maximum was achieved at the tip. Strain gauges could be placed in these two locations, and then the data compared with the simulation.

Along with the deflection of the blade, the weight of the blade can be measured and compared with the simulation. This could simply be done by lifting the blade onto large scales.

Knowing the true weight and deflection of the real blade, the stiffness and stiffness to weight ratio can be calculated and compared with the simulation.

As the **Nakashima Propeller Co (unknown date)** used to fully understand the deflection of the CFRP blade when in use, a full-scale propeller should be manufactured and mounted onto a shaft. The blade should be submerged in water that has a similar density to sea water. This can be done in a ship basin, as discussed in the previous report. Cameras could be set up to view the blade. When the blade is stationary an image can be taken from the camera, then another image can be taken when the propeller is rotating at different RPMs.

These images can be overlaid to note the deflection that has occurred at different situations, this would give a good indication of deflection and thus performance of the type of blade before retrofitting to a ship.

Technical literature and standards of a propeller

In the designing, building and operating of a CFRP propeller, there are multiple ISO standards that need to be met for commercial use of these designs. Below is a list of some of these ISO standards.

- ISO 484-1:1981 – Shipbuilding – Ship screw propellers – Manufacturing tolerances – Diameter greater than 2.5M
- ISO 13613:2011 – Ships and marine technology – Maintenance and testing to reduce losses in critical system for propulsion.
- ISO 20233-1:2018 – Ships and marine technology – Model test method for propeller cavitation noise evaluation in ship design
- ISO – 22098:2020 Ships and marine technology – Full scale test method for propeller cavitation observation and hull pressure measurement.

As the blade of the propeller in this design is over 2.5meter in diameter, then ISO484-1:1981 would need to be conformed with. This standard sets out the manufacturing tolerances of propellers above 2.5meters in diameter.

ISO 13613:2011 is for maintenance and testing of any components that relates to a ships propulsion system. The propeller is one of the most important components in the propulsion system, and maintenance of this would be critical. It would be important in this design to understand the maintenance requirements for this design to maintenance performance.

Cavitation of the propellers is an important factor to consider when designing ship propellers. Cavitation causes damage to the propellers and ultimately will affect performance. Therefore ISO 20233-1:2018 and ISO22098:2020 will need to be conformed to. These both relate to testing methods for cavitation, it is important to test for cavitation correctly and this could have an adverse effect on the design.

Conclusion

In conclusion using FEA optimisation in the design process is advantageous. FEA optimisation allows organisations to cut down design time, cut costs in producing multiple different prototypes and ultimately produce a superior product.

FEA optimisation has developed since its conception in 1940, and with computers becoming much more powerful and using cloud-based computing, then the number of processes that can take place with the FEA optimisation can increase, giving the results much quicker. And with the need to understand carbon footprint of products and reducing them as far as possible, software's such as this can help reduce material usage and possibly in future calculate the carbon footprint of these designs.

FEA optimisation does however require experience to use correctly. Mistakes are easily made, which could drastically affect results of design within FEA or in the real world. There is also a large cost to initially implementing FEA optimisation into an organisation.

For organisations to remain competitive in industry, they must invest in technology such as this.

Reference list

47.020.20 Marine Engines and Propulsion systems (2022) ISO. Available at: <https://www.iso.org/ics/47.020.20/x/> (Accessed: October 30, 2022).

Classnk (no date). Available at: https://www.classnk.or.jp/classnk-rd/assets/pdf/katsudou201511_D.pdf (Accessed: October 30, 2022).

Crahmaliuc, R. (2020) *75 years of the finite element method: Simscale blog, SimScale*. Available at: <https://www.simscale.com/blog/75-years-of-the-finite-element-method-fem/> (Accessed: October 30, 2022).

History of finite element analysis (FEA) (2022) O'Donnell. Available at: <https://www.odonnellconsulting.com/what-we-do/expert-witness-forensic-engineering/forensic-engineering-expert-witness/history-of-fea/> (Accessed: October 30, 2022).

Estimating propeller forces for Blade Strength Analysis - HydroComp (no date). Available at: <https://hydrocompinc.com/wp-content/uploads/documents/HC140-PropForcesForFEA.pdf> (Accessed: October 30, 2022).