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Structural Fluid & Analysis

Designing a tanker propeller

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Task 1

Finite Element Analysis is a very commonly used method to solve differential equations that can arise when designing engineering models. There are many techniques that can be used to support these problem areas. The problem areas that you could be looking to solve would be structural analysis, fluid flow and even heat transfer. Using your model or prototype you can simulate how these parts would actually perform in a real life environment. Using this technique you can prevent catastrophic failures and save time and resources making and test new parts constantly until you get the correct part. Putting these parts through testing can give you ideas on how much the part will weigh or how much stress or strain can this part go through before it fails. Finite Element Analysis was created in the 1950s in the aerospace industry. A professor at Berkley had written a paper on finite elements. In doing so Berkley then became the centre of research for FEA. A professor named E. Wilson created the first type of FEA software which was a massive hit in the 1960s. This software developed throughout the 1960s. Before FEA software was introduced a lot of it was done manually. They would have used the earlier electronic calculators and computers but a lot would be done by large scale testing and manually working out how the part would perform. Also before this software analog computation provided the alternative from hand calculation. These analog computers were used to simulate numerous physical processes. Before the computers and software we had to day employers would have a maths department where the only job the people had were to mathematically calculate like inverting matrices. This would be a service to the other technical departments.

In Finite Element Analysis you have Static and Dynamic structural. Static analysis is when a load is applied very slowly to a part so that you can see the structure deform very slowly. This shows how the part will perform without any vibrations or big impacts. Dynamic analysis is the opposite. This will simulate the part under high vibrations and impact which creates a lot of inertia. Both of these analyses have linear and non-linear types of analysis. Some types of non-linear analysis would be contact non-linearity. This would arise when the boundary conditions of the part you are modelling would change during your analysis. Material non-linearity is another form of analysis, and this is where the stress overcomes the yield stress. This is when the material properties change with applied loads.

Dynamic Analysis has many forms of analysis. One of these techniques is model analysis. Model analysis determines the mode shapes of the structure. Another good technique would be response spectrum analysis. This is when you study the response of the structure when a load is applied to it.

Using simulation is key to a an efficient and cheaper design. Simulation allows the engineers to test any assumptions or theories that they have come up with. By doing this there is no need for the expense of material and manufacture of a physical prototype. Especially with the standards that need to be met a physical prototype would be very costly to a company. The future of FEA will only mean more benefits for the user. New numerical techniques could be used that will make the result more accurate to real life applications. It will also be established as a very important tool for the design and production of innovative and efficient models without room for error.

Task 2

Using a blade that was created in solid works I imported this into Design Studio. The blade that was imported in was designed as a steel blade. From assignment 1 we had come to the conclusion that a carbon reinforced blade would be more appropriate. I made some assumptions. These assumptions are using the first assignment as help. I can assume that the blade will be submerged in sea water which has a density of 1023.6 KG/m³. I can also assume that an average speed of a tanker ships propeller is around 245 rpm. 245 rpm is around 25.6 m/s. Using these assumptions, I can calculate a rough figure of the force.

To calculate the force, I used $F = m \times a$.

$$F = 32\text{kg} \times 25.6 \text{ m/s} = 819.2 \text{ N of force.}$$

I then divide this by the number of nodes so that it is a Uniformly distributed load.

$$4848 / 819.2 = 5.91 \text{ KN of force on each node.}$$

So from this I had to change the material of the part. The materials properties that I had put in are shown below.

Young's Modulus in the longitudinal direction (E_1):	132 GPa
Young's Modulus in the lateral direction (E_2):	10.8 GPa
Poissons Ratio (Nu_12):	0.24
Inplane Shear Modulus (G_12):	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/m ³
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 inplane shear stress (S):	43.4 MPa

In the Fig.1 below you can see me inputting the data into the design studio software.

Materials

< Back
Next >
Finish
Cancel

Edit a Material : 2-D Orthotropic Properties

E_1:	132.0
E_2:	10.8
Nu_12:	0.24
G_12:	5.65
G_1z:	5.65
G_2z:	5.65
Rho:	1540.0
A_1:	
A_2:	
Tref:	
GE:	
F12:	

☒ Stress Limits
☐ Strain Limits

Xt:	1513.0
Xc:	1000.0
Yt:	43.4
Yc:	43.4
S:	43.4

Fig.1

After I inputted the material properties, I need to then orientate the elements that are relative to a vector which the material property uses. By generating these THETA vectors, they default to aligning to 0 degrees on the X axis. We have assumed that the vectors need to be aligned to the y axis at 0 degrees. In Fig.2 I have shown how I have generated the Theta vectors. The vectors are in the default alignment and in Fig.3 I have shown when it was modified.

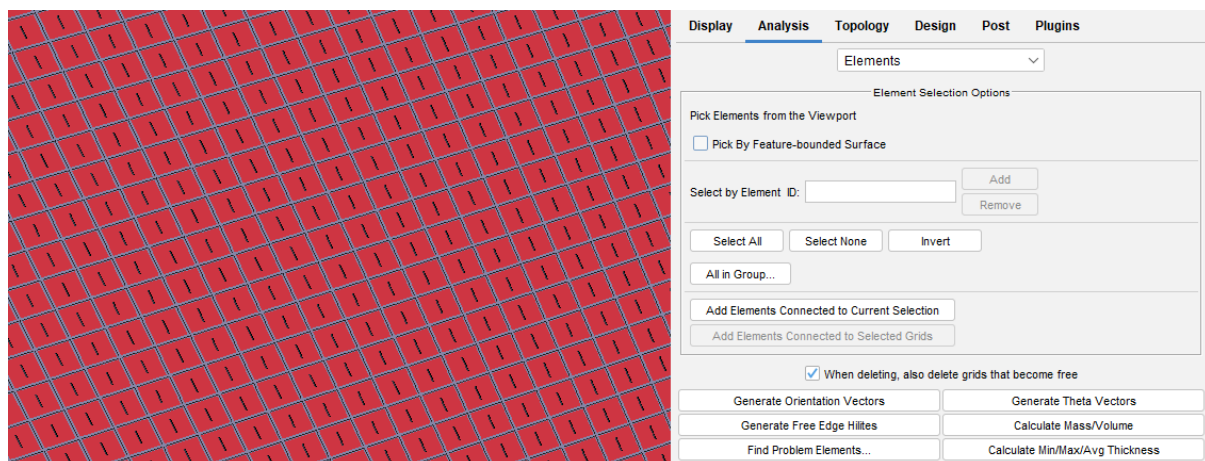


Fig.2

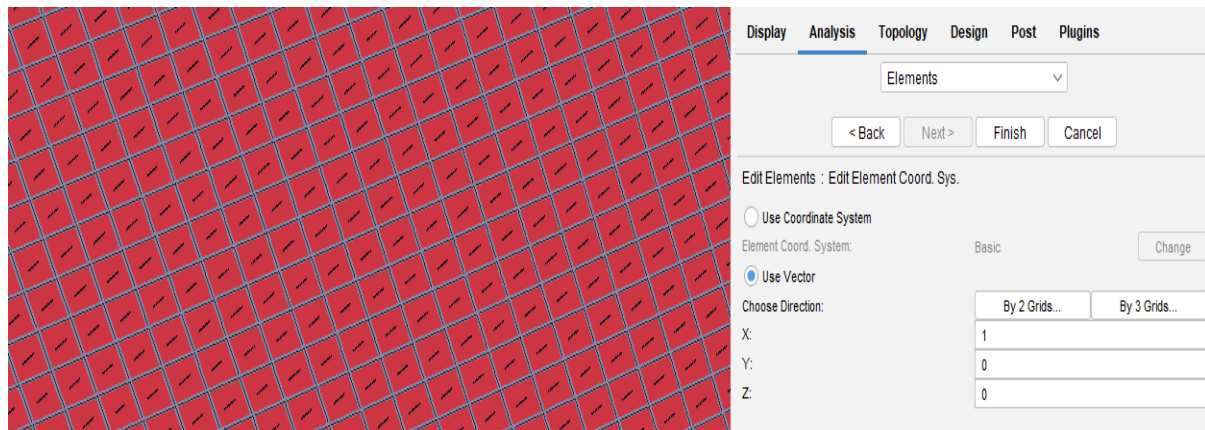


Fig.3

Now that this is complete we are going to make this into a material that we have defined. After doing this I need to compile my composite. I do this by adding 25 plies for the UD. I laid 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. You can see this in Fig.4 below.

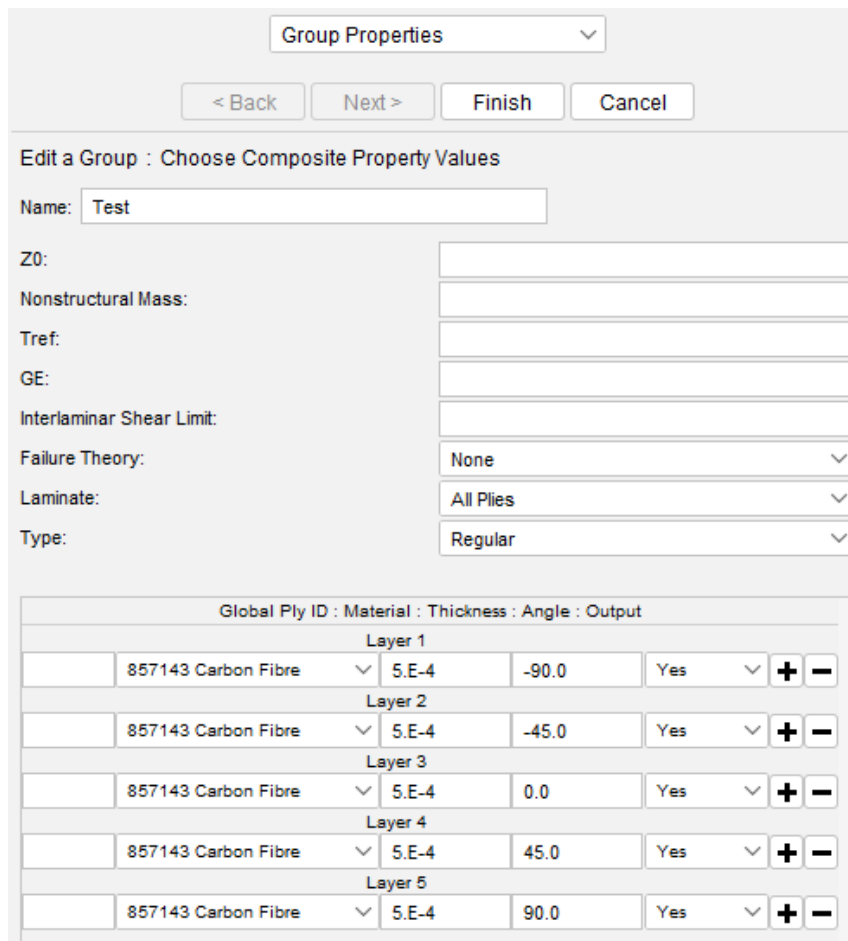


Fig.4

I can now assess the blade by using a single analysis inside the software. In Fig.5 below you can see the single analysis that had taken place.

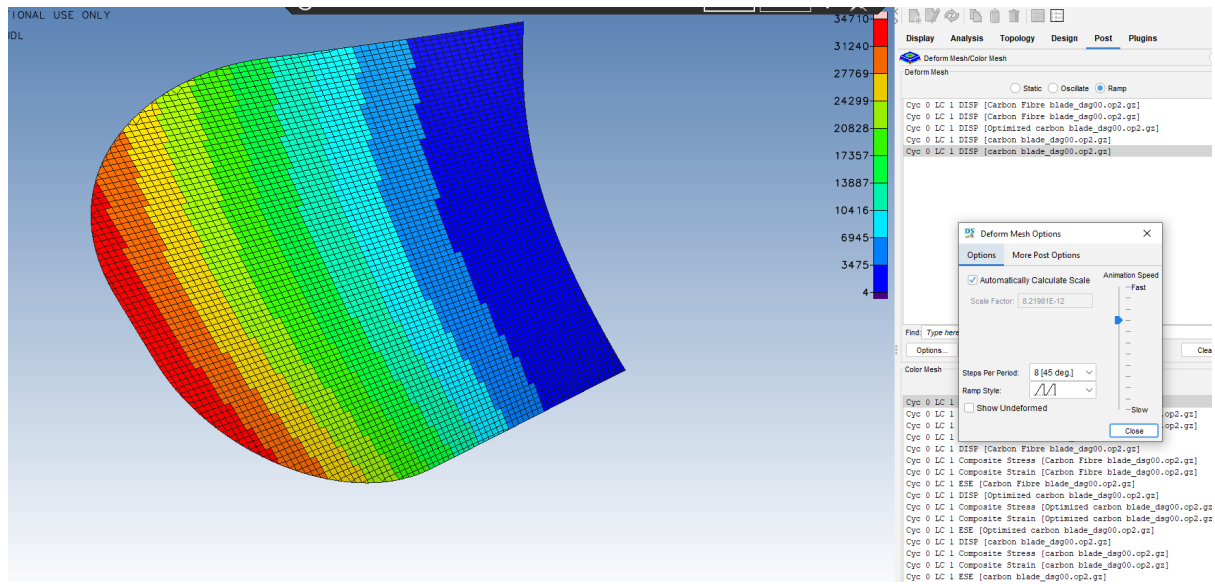
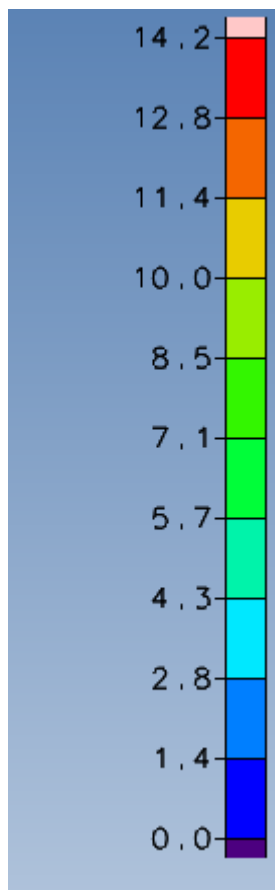


Fig.5

After assessing this I can now see the results of my displacement and mass.
My displacement as you can see in Fig.6 is 14.2 mm and my mass is around 32kg.



I then used the software to get all the calculations that I needed. Before the carbon composite I did a steel blade test so that I can compare afterwards.

Run	Material	Thickness	Deflection	Mass	Load	Stiffness	Stiffness/Weight
Steel Baseline	Steel	3mm	0.22mm	646kg	100N	227.27 N/mm	358 N ² / mm
Composite Baseline	Carbon	5mm	14.2 mm	32kg	100N	35.2 N/mm	1121 N ² / mm

To work out the stiffness for steel and carbon fibre I used the formula below.

$$K = f/x = 500/0.22 = 227.27 \text{ N/mm (Steel)}$$

$$K = f/x = 500/14.2 = 35.2 \text{ N/mm (Carbon)}$$

To work out the stiffness to weight ratio I used the following equation

$$K = k/(mg) = 227.27 / (646 \times 9.81) = 0.0358 = 358 \text{ N}^2 / \text{mm}$$

$$K = k/(mg) = 35.2 / (32 \times 9.81) = 0.0112 = 1121 \text{ N}^2 / \text{mm}$$

In terms of stiffness to weight ration the composite part really doesn't have any advantage over the steel structure. Hopefully by optimising the structure it will be more accurate and give us better results over the steel blade.

Task 3

Now that I have completed the material change from steel to carbon. I feel that the carbon composite model can be improved. I am going to do this by optimising the model. By doing this we will be hoping to achieve the same deflection as the steel structure but at a reduced weight.

To do this I saved another copy of my carbon blade. I then used the design tab in design studio to set up the constraint. I set up constraints so that the thickness of each orientation can be between 0.2mm and 3mm. As you can see in Fig.7 this is where I set my upper and lower values.

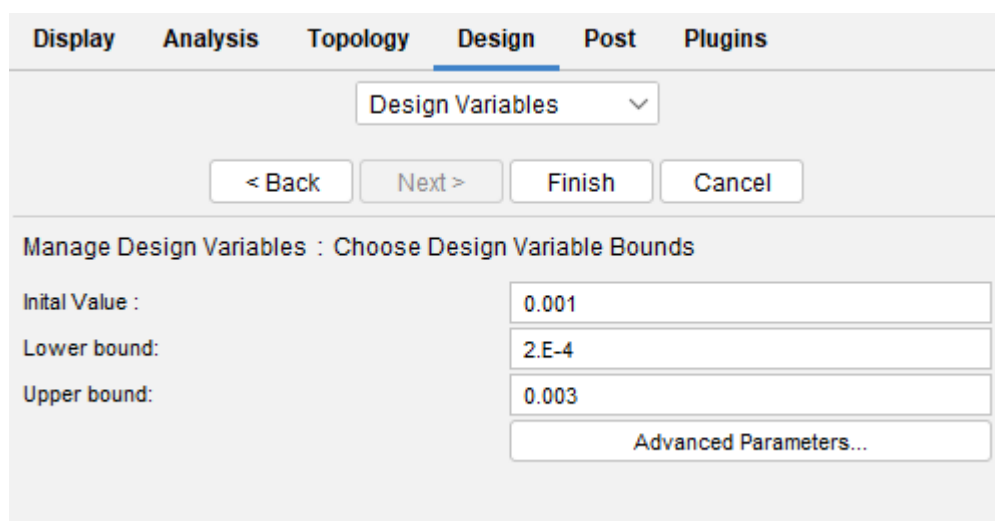


Fig.7

I then added 5 of these which will represent the variables of the thickness of the Plys. You can see this in Fig.8 which is below.

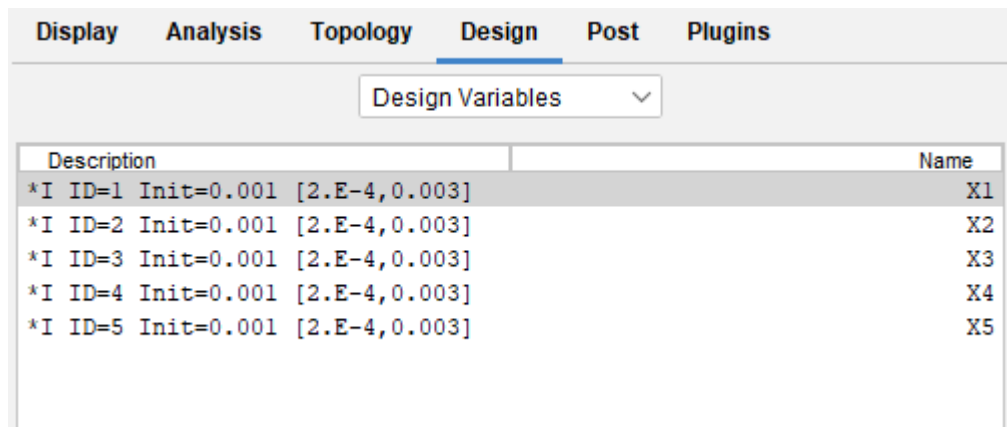


Fig.8

After this I can now attach my design variables to my PCOMP that I created earlier.

After doing this I can now optimise my blade and gather the data and results. Here are the results from the first optimisation. As you can see in Fig.8

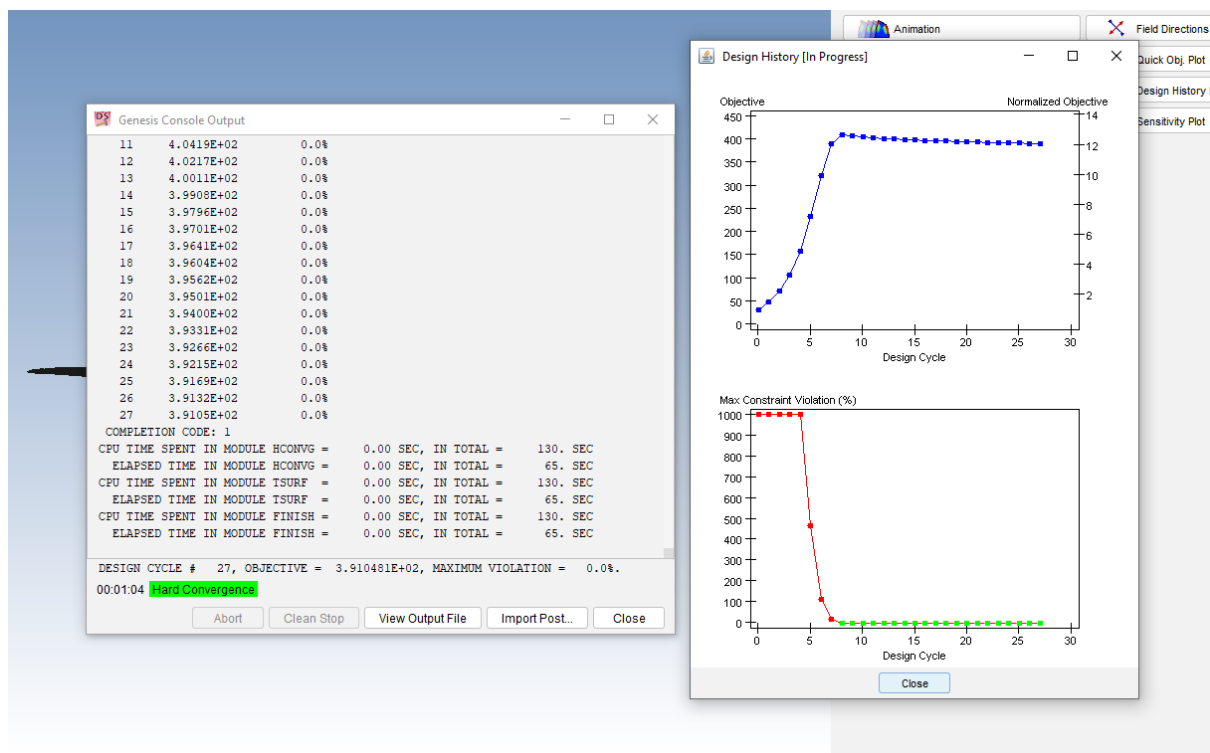


Fig.9

As you can see in Fig.9 above. In the design I am violating my constraint at around 1000% but meeting my displacement constraint maximum at around 12mm.

Genesis had added thickness to each orientation so that it meets the displacement target which in turn till minimise the mass. By looking at how the variables have changed we can see that all variables started at 2.4e-3 but my plys at 0 degrees have increased to my maximum of 3 mm which is what you would expect. My plys at -90 and -45 have then reduced to my minimum and my plys at

90 and 45 were approximately 3.9×10^2 mm. These to real world applications are not very accurate but due to it being a simulation and maybe error on my side I don't get an accurate figure to my stiffness target.

I then generated and added 14 additional values to all 5 of my design variables. Which you can see in fig.10 below.

Fig.10

I then ran a genesis optimisation cycle again and this time my stiffness target had increased to 4.074×10^2 mm. I also got a fatal error around design cycle 29. As I said before due to my error or the fact that it is a simulation and I might be getting programming / software which is giving me a fatal error. Fig.11 below shows my fatal error.

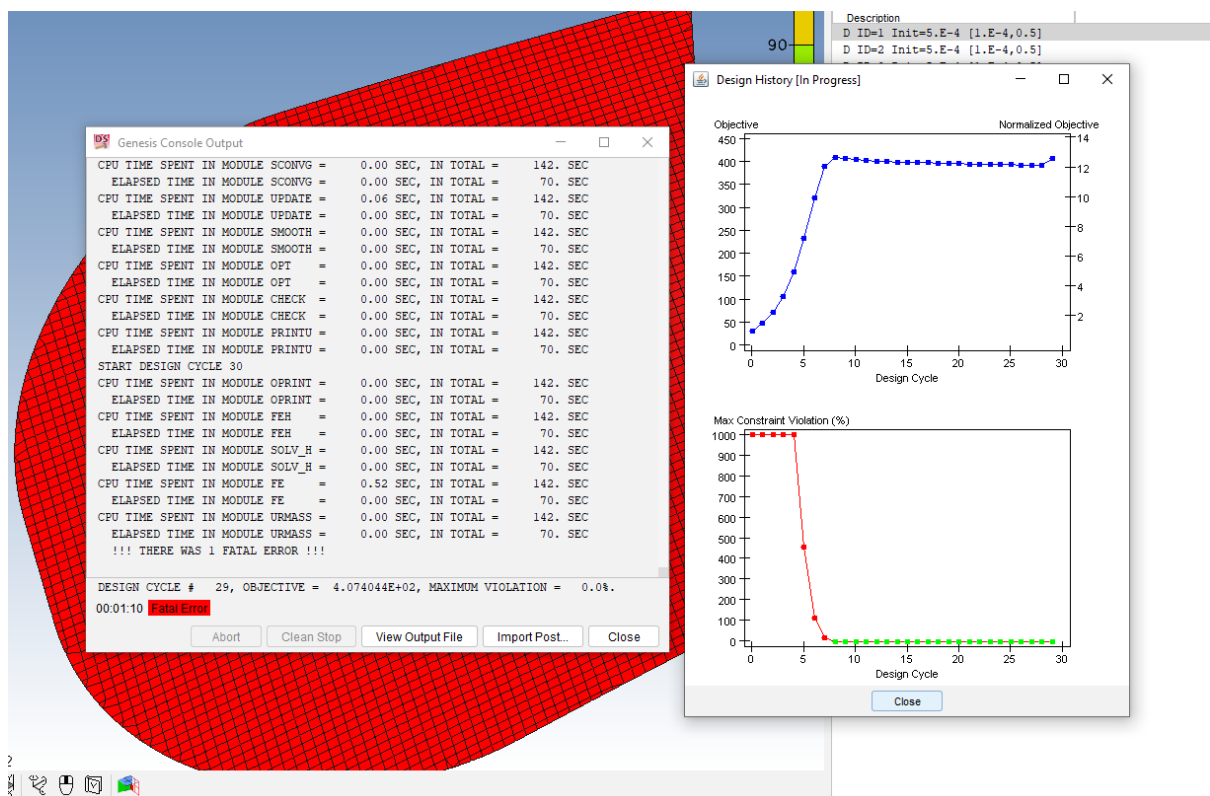


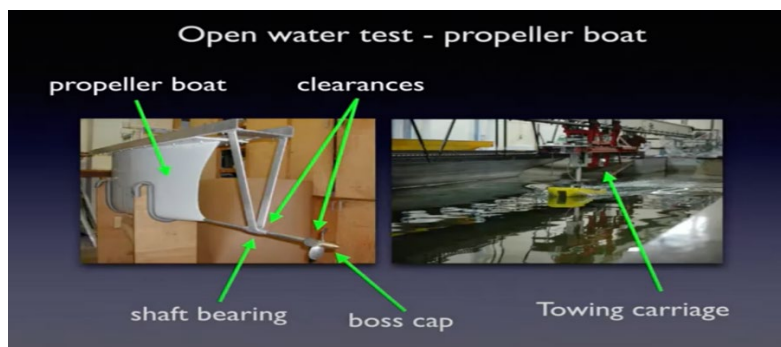
Fig.11

My weight was still around 32kg but theoretically it will still be more efficient as theoretically I am getting a stiffness to weight ratio of around $1121 \text{ N}^2 / \text{mm}$ compared to the $358 \text{ N}^2 / \text{mm}$ that I get for the steel blade.

In conclusion I believe that theoretically a carbon reinforced blade will add a lot more benefits and efficiency compared to the steel blade. The weight saving on the part and then the stiffness to weight ratio is where the efficiency of the carbon reinforced blade really shows the optimisation of using a different material. The carbon reinforced blade allows a lot more flex compared to a steel blade that under high flex and stress will start to break down and gather a lot of wear and tear. There are benefits to both blades but a steel material is often used due to the fact that they are corrosion resistant materials. This is a big factor has the propeller will always be submerged in water. However, A polymer reinforced carbon fibre blade can be one of the best corrosion resistant material available increasing the longevity of the blades. A reinforced carbon fibre is very lightweight and can flex a lot. This will help with the flow of the blade propeller compared to the steel blade cannot flex and bend as much which we have proven in the modelling techniques that we have used,

Task 4

To physically test a prototype propeller I suggest we do an open water test. An open water test can be done in a towing tank or a cavitation tunnel. The test is normally not dimensioned and is used alongside a resistance test. By testing this propeller in a confined environment, you can collect data on its thrust, torque and the performance characteristics of the propeller. A towing tank which you can see in the image below will have a carriage that will pull the ship or the propeller along the channel. On this carriage we build a propeller board that acts as a form of propulsion due to its shaft that will turn the propeller. Compared to a cavitation tunnel where the propeller stays at a fixed point and the water then moves around the propeller. I feel that the open water test has a few downfalls as you constantly have to wind back the carriage and wait for the water to settle before trying again. In the cavitation tunnel you are able to constantly monitor it and change speeds whenever.



Ref 1

By varying the speed of the carriage and the RPM of the shaft we get all the parameters from the propeller using a dynamometer that calculates the torque and rotational speed of a motor. It also will display to us the thrust. The data we want to gather from the test is the torque and thrust in relation to the RPM. To do this test we should run it at different speeds and gather the data at each speed from the dynamometer. By using the coefficient we are able to non-dimensionalise the data. The thrust and torque are taken at the coefficient so that it is easier to scale for bigger propellers. Running the test at a low RPM will give us a high value whereas a higher RPM will give us a lower value. As you can see from the image below you can see how the propeller acts in the water.



Ref 2

By doing these tests we can determine whether the blades are aligned correctly or if the pitch is designed correctly.

I believe that comparing these tests depends on who is undertaking the studies. As shown in my modelling there is room for a lot of errors when undertaking these studies. A engineer who has spent a lot of time on this software could easily make the simulation work to where it is as close as possible to real life application. The time saved on just simulating the part could in the long run save a company a lot of money when proposing a design for the blade. Using simulation and then creating a part. Running these tests to compare whether the simulation results and the real life testing results would also be a very helpful in understanding if your simulation was correct. I believe these tests should be used but the simulation should be used early on in the design process to understand and pick up on anything that could cause a catastrophic failure of your blade.

Task 5

When designing a propeller blade for a ship there will be standards that are already set to ensure they are safe. Like most products and engineering parts standards are there to ensure customers that the engineers behind the design have met these standards and that the part will work in society's best interest. By meeting these requirements the products going out are safe and can be used by the public safely.

When designing a propeller blade there are already a lot of standards that are set. Some of these standards are that the leading edges of the blade require a very high level of accuracy. This is to ensure that when going through water the blade is smoothly going through and not creating a lot of load and force on that blade.

Taken from Ref 3 "A tolerance within 0.05 mm is recommended for the edge sectional shape (leading, trailing, and tip edge geometry)." So like I have just explained this standard is set as this across the board works best for a propeller blade. Also when locating all the blades they should all be equispaced properly or this will cause failures. The standard for these between two consecutive blades is plus or minus 1°. Which is a very tight tolerance to adhere to. Another big part on how efficiently your blade will perform is the surface finish of the blade. Carbon reinforced plastic will already have a very smooth surface finish compared to steel blade that will have to go through some sort of polishing process. Using ref 3 again you can see that what I have just said is stated in standards for these propeller blades, "The model propeller should be polished and inspected for blade surface blemishes, which could affect cavitation performance. The mean roughness of the

blade surface should be in the range of 0.3 - 0.5 μm " A lot of these standards are mainly just specifying the tolerances that you should adhere too. For example, the diameter of a blade will vary depending on the size of the ship that you are working on. Using the simulation techniques that I have shown in the early part of the assignment. When you get a design that on a simulation works very well when producing that part you will have a tolerance to ensure that the part you have created meets that of the simulation. The standards set for the diameter should be within $\pm 0.05\%$ of the design diameter with a maximum tolerance of $\pm 0.1 \text{ mm}$. These are very tight tolerances but when producing high level parts accuracy is key. When moving away from these tolerances you will notice big performance issues and even failures on your parts.

References

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