

CAE Assignment

by Finn Mallon

Submission date: 01-May-2023 05:34PM (UTC+0100)

Submission ID: 204140798

File name: 33170_Finn_Mallon_CAE_Assignment_497865_1316576056.docx (2M)

Word count: 2639

Character count: 14101

University of Wales Trinity Saint David

BSc Ordnance, Munitions & Explosives



Application of Computer Aided Engineering Processes

Computer Aided Engineering (ACDB5007)

Finn Mallon

2031021

Date (05/2023)

Lecturer: Andrew Noble

Contents

1.0	Introduction	3
1.1	Aims & Objectives	3
2.0	Background	3
2.1	Advantages and Dis-advantages	3
2.2	How CAE Is Used In the OME Industry	4
3.0	Methodology.....	5
3.1	Solid works Steps -	5
3.2	Ansys Steps –	6
4.0	Validation	7
		9
6.0	Conclusion.....	13
7.0	References	14

1.0 Introduction

This report will give an introduction to how CAE has become a significant part of engineering over the last few decades. Explaining how it can be used to assist in test and design of component development but also its limitations with regards to other possible Test and design routes.

The report will also give an example model showing the different components used as well as showing the kinematics analysis of the system under realistic operating conditions as well as a static structural assessment of one component within the assembly.

1.1 Aims & Objectives

- Critically analyse and evaluate techniques for structural and fluid problems and CAE results.
- Demonstrate competence in the application of suitable software packages to investigate engineering structural and fluid problems and cognitive skills in the formulation and analytical solution of a problem within the chosen engineering specialism.

2.0 Background

CAE is a widely used practise in all industries that require some element of design. It has many benefits that prove that it is definitely the way forward for design and simulation however, there are also some downsides that may affect the idea that it is a beneficial way of doing things. Some of these advantages and disadvantages are listed below:

2.1 Advantages and Dis-advantages

Advantages:

- **Saves Time.** When doing the initial drawings CAE will save lots of time. This is vital in industry and allows the team to meet more deadlines to keep the customers happy. It is a lot simpler and less time consuming to create a drawing using the software than it would be using old methods of drawing boards.
- **Easy to Amend.** Making alterations in designs and simulations is essential and having the ability to do this using CAE is a great benefit. Customer requirements may change, meaning modifications to the design may need to be made ultimately effecting how the simulation runs, leading to different results.
- **Decrease in Error Percentage.** CAE software is very accurate and shows the measurements and parameters on screen meaning the user can easily see if they have inputted the correct Information. CAE software also allows the user to copy various different parts to enable no differentiation between each one allowing for more accurate designs and simulations.
- **Decrease in Design Effort.** The amount of effort required to create a model and to simulate the results of different affects has significantly decreased. This is ultimately down to having the option to automate certain parts of the design and copy and paste parts that have been

made. The simulation aspects also requires no effort from the user apart from initiating the simulation.

- **Easy to Share.** Saving and storing the files that have been created and having the ability to share with anyone is another great benefit of having CAE. This allows different users to all have access to different files and to be able to amend them when necessary.
- **Improved Accuracy.** The accuracy within CAE is second to none and cannot be replicated by any other means of design. The tools within the various different software packages is so precise and will only allow the user to use certain parameters if it is physically possible to do so. [1]

Dis-advantages:

- **Easy to lose.** Work can be lost easily if the computer decides to malfunction or breakdown whilst a task is being carried out. Although not very common nowadays this is always a possibility and shows how important it is to save work on a regular occasion.
- **Easily Hacked.** Computers can be hacked and corporations are big targets for hackers trying to get information.
- **Learning to Use Software.** Learning to use the CAE software is very time consuming as there is a lot to learn.
- **Simulation.** The simulation aspect of CAE is very time consuming if there are lots of different parts and parameters involved.
- **Price.** CAE Software can be very expensive depending on the type of software package that is purchased. There are many different packages available some cheaper than others. [1]

In my opinion the advantages that CAE brings to the table out way the small amount of disadvantages. Most of the disadvantages listed above can be easily avoided with the correct precautions in place. For example, (Easy to lose), how often does a modern computer these days just magically decide to malfunction. This is very uncommon and the chance it will happen to a company that uses CAE that doesn't have the files backed up is minuscule. Another example, (Easily hacked), yet again having the correct precautions in place such as using a firewall or having anti-virus protection will reduce the chance of this occurring.

2.2 How CAE Is Used In the OME Industry

CAE is used in lots of different industries, some of the most popular ones are:

- Medical
- Civil Engineering
- Automotive
- Architecture
- Aerospace [2]

All of these examples will you CAE in different ways and for different purposes. This is another great advantages of CAE is that there are no limitations on what it can be used for.

CAE has a long history of different uses however, according to multiple articles online, CAE was initially created in 1943 with the first general purpose computer by the combined effort of the US army and the University of Pennsylvania. It was created and used to calculate ballistics for US army artillery shots. [3]

3.0 Methodology

The first step to any CAD / CAE project is engineering drawings. Before online drawings were a thing all engineering drawings were drawn out by hand using drawings boards and a straight edge.

Nowadays all engineering drawings are done on the computer which makes it a lot simpler and easier to understand. Engineering drawings can be as simple or as complex as the creator wants them to be however, having more detailed displayed on them will mean for an more accurate set of results at the end. Although, having too much detail can become confusing and in some cases the program being used to obtain the results may not let the user do such things. It's all about having that happy medium of parameters set. Once the engineering drawings have been obtained from the customer then the following steps can be made to start to create the desired result:

3.1 Solid works Steps -

- **Create the individual parts.** On a suitable program such as (Solid works) make sure that all of the measurements are accurate and match whatever the measurements are on the engineering drawings.
- **Assembly.** Once all of the individual parts have been created they can start to be assembled together. This can be done using fasteners such as bolts or screws. Having bolts or screws in place will allow the two components to be able to be separated when a prototype model is made however, if no fasteners are used then within the CAD packages there is a tool called 'mate' this will attach separated parts to one another and they will not be able to be separated once the prototype mode is made. This is done by selecting the separate faces of the different components then by selecting the mate tool, which will join the part together.
- **Test for interference.** This will test for interference between components, which will help you to examine and evaluate those interfaces. This is very useful in complex assembly's where it is difficult to visual determine whether components interfere with each other. [4]

- **Check for Errors.** A list of errors that have been made can be selected and brought up by clicking on the feature menu design tree. This will display all errors and warnings where the user can work through them one by one. [4]

3.2 Ansys Steps –

- **Import into Ansys.** Once the components have been made into an assembly it can then be imported into Ansys for the finite element analysis (FEA). This is where different material properties can be added to the assembly and different tests can be carried out on the component such as static structural studies, thermal studies or rigid dynamic studies.
- **Mesh model.** Once the component has been imported into the Ansys system a mesh needs to be applied. The reason for this is that it turns any irregular shapes in the assembly into more recognisable volumes named elements. Having the meshing applied will significantly reduce the time it takes to obtain the results. However, meshing is a required a decision to be made on what size to use. Having the meshing too fine will mean obtaining the results from the analysis will take far too long. Having the meshing too large will mean crucial data will be missed from the analysis. [5]
- **Assign Material Properties.** Next step is to decide what materials the component needs to be made from. Material data packages will help with this such as (Granta Edu pack), these packages know all about the different materials and there properties such as the young's modulus of elasticity, Poisson's ratio, density, tensile yield stress and much more. This makes results much more accurate as the analysis takes much more material properties into account.
- **Define boundary conditions.** Once the material properties have been identified and selected to the component then boundary conditions can start to be applied. For example if a part of the component is going to withhold a load or then this is a boundary condition that will be applied.
- **Setup analysis.** What type of analysis is going to be carried out? Is it a static structural analysis? A static structural analysis determines the steady loading conditions on a structure whilst omitting inertia and damping effects such as those caused by time varying loads. [6]

Is it a Thermal study? Thermal study is where analysis is carried out to show the time and temperature at which physical changes occur when the component is heated or cooled. [7]

- **Solve the problem.** Once all of the material properties have been entered, all boundary conditions have been applied and a mesh has been created the solve button can now be used to run the analysis to gain a set of results. The user may incur some problems at this stage however, these can be easily worked through by accessing the feature menu design tree.
- **Post process the results.** This is an efficient way of validating the results of an analysis. Post processing can be done within Ansys, where the general post processor is used to review the entire model in one moment in time or frequency. The time-history processor is used to review the output at specific locations through the time of frequency range. [5]
- **Validate and refine model.** Once all results have been obtained it is important to validate the results. Some standard approaches to validating an FEA process:
 - Hand calculations
 - Comparison to standard results
 - Comparison to physical data
 - Comparison to other computational simulations

The model can be refined to ensure accuracy if there are any discrepancies between physical data evidence and other computational simulations.
- **Report findings.**

4.0 Validation

Validation is the process of assigning a quantifiable value to the accuracy of a computational simulation relative to the real-world scenario. This incorporates the model, simulation and results output.

Finite Element Fundamentals (FEA) – Involves breaking down complex scenarios into smaller, manageable ‘finite elements’ which can be solved easily.

The finite elements are interpolated representations of the real world scenario that can be represented by mathematical equations and solved computationally.

Accuracy of the representation will affect the accuracy of the results.

Some standard approaches to validating an FEA process:

- Hand calculations
- Comparison to standard results
- Comparison to physical data
- Comparison to other computational simulations

The following Non-linear validation study example is what I will be using to set my material properties and boundary conditions.

Verification Example

Program: RFEM 5

Category: Geometrically Linear Analysis, Isotropic Linear Elasticity, Plate

Verification Example: 0016 – Elastic Bending - Continuous Load

0016 – Elastic Bending - Continuous Load

Description

A thin plate is fully fixed on the left end ($x = 0$) and is subjected to a uniform pressure p according to the Figure 1. The problem is described by the following set of parameters.

Material	Elastic-Plastic	Modulus of Elasticity	E	210000.000	MPa
		Poisson's Ratio	ν	0.000	
		Shear Modulus	G	105000.000	MPa
Geometry	Plate	Length	L	1.000	m
		Width	w	0.050	m
		Thickness	t	0.005	m
Load	Plate	Pressure	p	2750.000	Pa

Small deformations are considered and the self-weight is neglected in this example. Determine the maximum deflection $u_{z,max}$. The aim of this example is to show that a surface of the surface stiffness type *Without Tension* behaves linearly under bending.

Figure 1: Problem sketch

Analytical Solution

The bending moment M for the plate under continuous loading $q = pw$ is defined by

$$M = -\frac{q(L-x)^2}{2} \quad (16-1)$$

Verification Example - 0016 © Dlubal Software 2016

0016 - 1

[8]

Verification Example: 0016 – Elastic Bending - Continuous Load

Recalling the elementary equation for the beam deflection using the Euler-Bernoulli theory

$$EI_y \frac{d^2 u_f}{dx^2} = M \quad (16 - 2)$$

where I_y is the quadratic moment of a cross-section with respect to the y -axis and it is given by

$$I_y = \frac{wt^3}{12} \quad (16 - 3)$$

We integrate the equation (16 – 2) from 0 to L twice. Upon applying the boundary conditions for cantilever beam $u_f(0) = u_f'(0) = 0$, the solution takes the following form

$$u_f(x) = \frac{qx^2(6L^2 - 4Lx + x^2)}{24EI_y} \quad (16 - 4)$$

The maximum displacement occurs at the point $x = L$:

$$u_{f,max} = \frac{qL^4}{8EI_y} = \frac{3pL^4}{2Ew^3t} = 157.143 \text{ mm} \quad (16 - 5)$$

RFEM 5 Settings

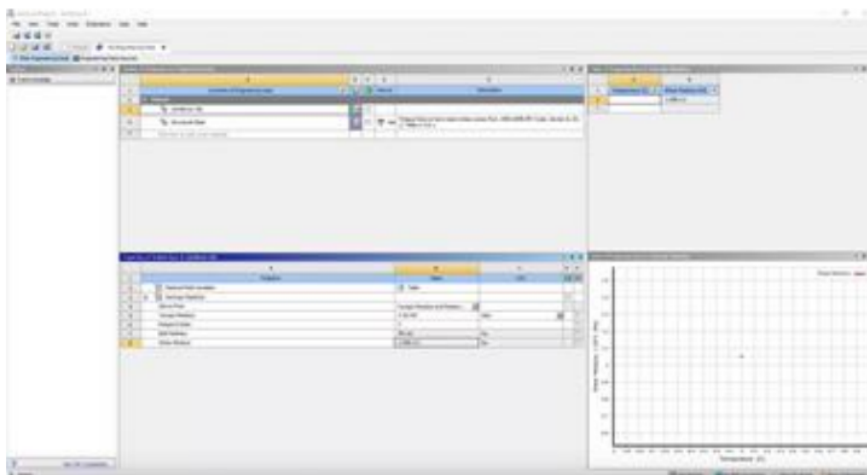
- Modeled in RFEM 5.04.0024
- The element size is $l_{el} = 0.020 \text{ m}$
- Geometrically linear analysis is considered
- The Mindlin plate theory is used
- Isotropic linear elastic material model is used

Results

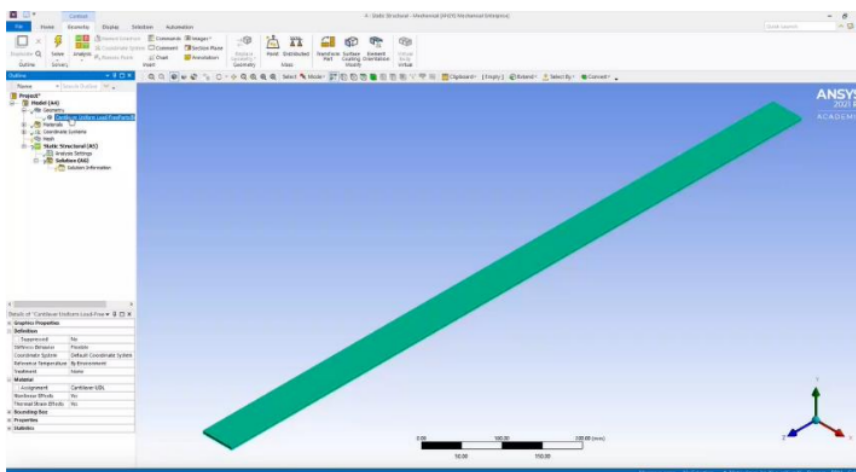
Structure File	Surface Stiffness Type		
0016.01	Standard		
0016.02	Without Tension		
Surface Stiffness Type	Analytical Solution	RFEM 5	
	$u_{f,max}$ [mm]	$u_{f,max}$ [mm]	Ratio [–]
Standard	157.143	157.130	1.000
Without Tension		157.150	1.000

[8]

The analysis that will be carried out is Static structural. The first step is to enter all of the correct Engineering data that matches the validation study example above. As you can see from the image below I had created a new material type called 'Cantilever UDL' then inputted all of the matching data. Setting the 'derive from' tab to youngs modulus and poisons ratio then changed the youngs modulus value to 210,000MPa and Poissons ratio value to 0.00. The system then updated the Shear modulus value based off my two inputted value to 105,000MPa.

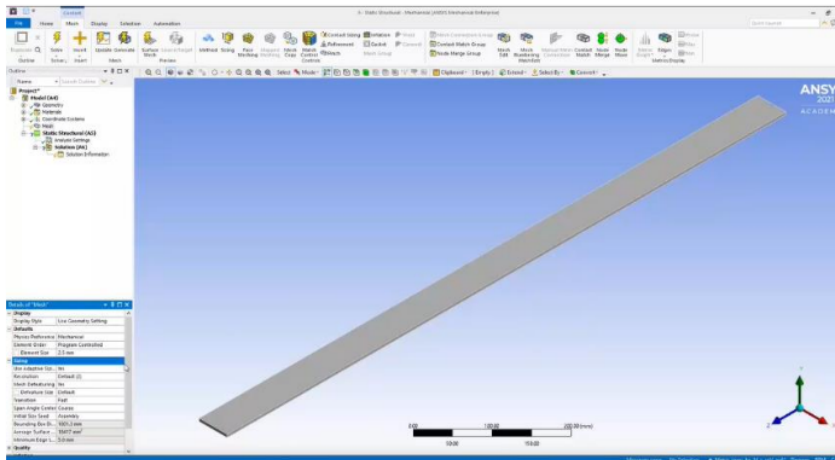


[9]



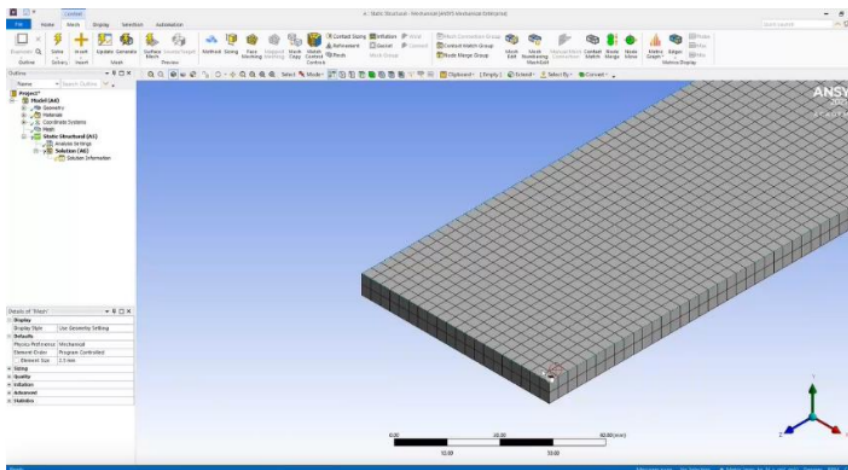
[9]

The next step is the geometry. I firstly made sure that the Stiffness behaviour was set to 'flexible'. Then changed the material to the Cantilever UDL material that was just created.



[9]

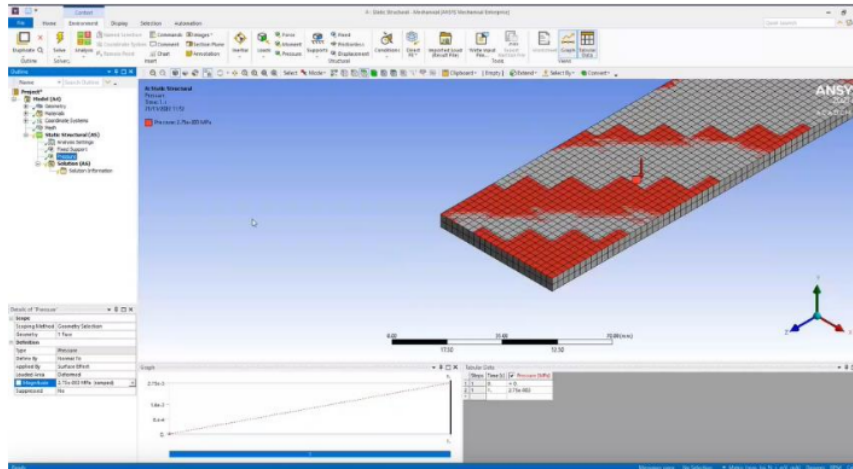
The mesh was the next step. This is very important and getting the mesh size correct will have a big impact on the results gathered at the end. I changed the mesh element size to 2.5mm, then changed the transition to slow to make the transition from meshing larger elements and smaller elements a lot smoother to avoid any problems. I then ran the mesh and let the program do its thing.



[9]

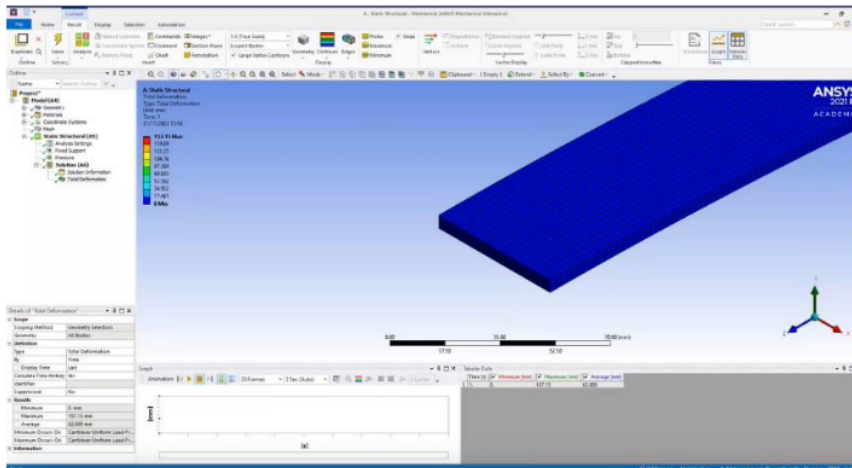
As can be seen from the image above there were no problems with the mesh size that was chosen and it covered the component well.

I then applied a fixed support on the end face of the cantilever. Then I applied a pressure load which was uniformly distributed across the whole of the top surface of the cantilever with a magnitude of $2.75\text{e-}3$ to match the load value on the validation example. As can be seen on the image below.



[9]

The analysis was now ready to solve. As can be seen from the solve button which is displayed in the top left hand corner of the image above. However, before the solving could begin the solution needed to be set to Total deformation to yet again match what is shown on the validation example. I then ran the analysis to see if the same results as the validation example would be obtained. Which can be seen on the image below.



[9]

As stated in the validation example the Total Deformation was 157.143mm which was the same result obtained from test I have carried out.

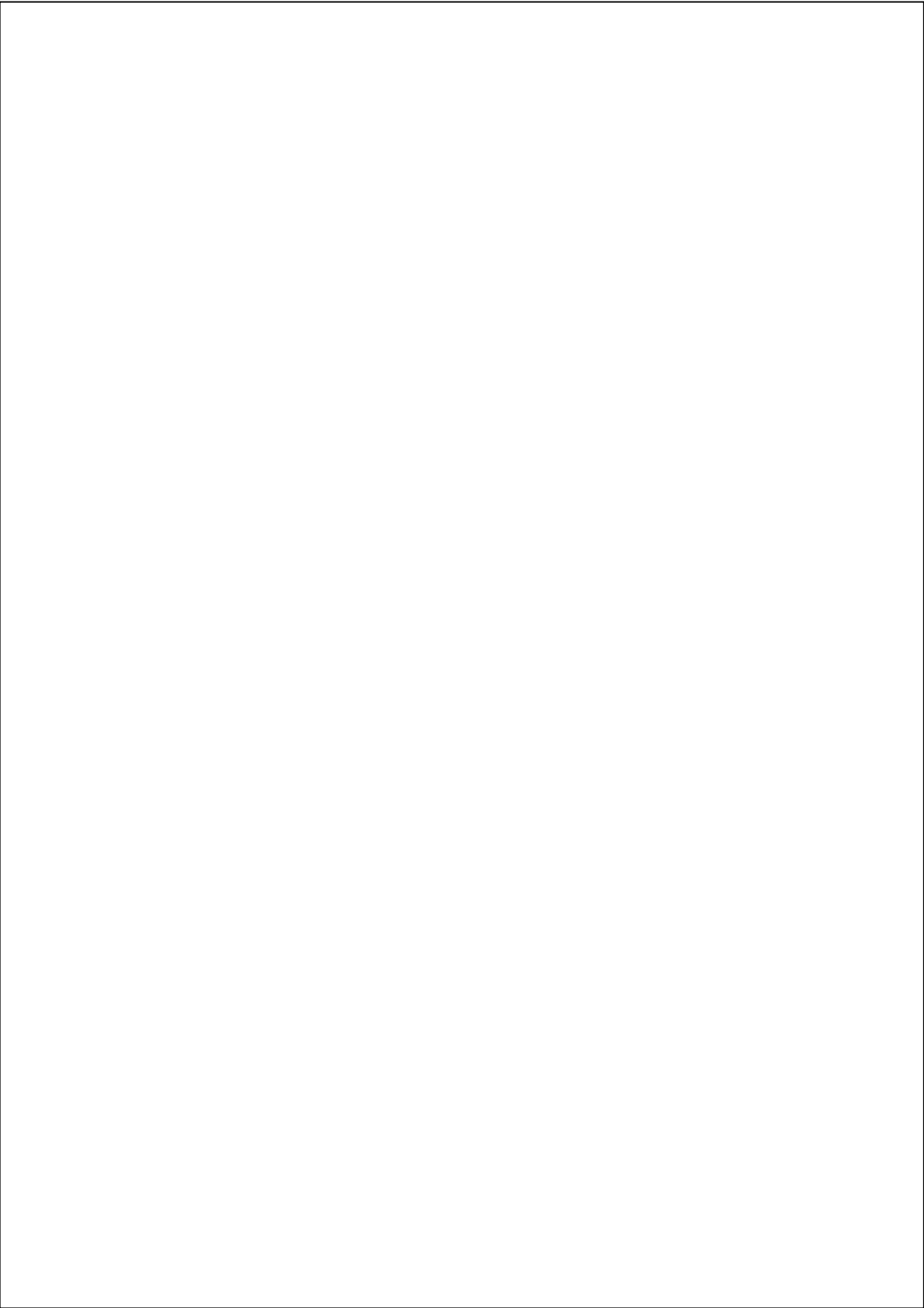
6.0 Conclusion

After carrying out the static structural test using a validation study example and getting to use CAD as part of CAE it has been proven to me that all of the initially stated advantages within the introduction to this report ring true. I have found that it is very accurate and easy to use. It is easy to amend if problems occur and there are lots of tools to use that will make the user job a lot simpler.

Overall from a business stand point I felt it is a very beneficial program to have in place and will save a lot of time and effort as well as ultimately saving money. The FEA system is great and I can understand why it is used in many different industries across the globe.

7.0 References

- [“ARCVERTX,” 2018. [Online]. Available: <https://www.arcvertex.com/article/advantages-and-disadvantages-of-using-computer-aided-design-cad/>. [Accessed 20 04 2023].
1]
- [“CAD/CAM Services,” 22 December 2020. [Online]. Available:
2 <https://www.cadcam.org/blog/many-uses-cad-cam-software/>. [Accessed 22 April 2023].
3]
- [“NOVA,” [Online]. Available: <https://nova-design.co.uk/history-of-computer-aided-design/>.
3 [Accessed 22 April 2023].
4]
- [D. Systemes, “Dassault Systemes,” 2021. [Online]. Available:
4 https://help.solidworks.com/2021/english/SolidWorks/sldworks/c_interference_detection.htm#:~:text=Interference%20Detection%20identifies%20interferences%20between,components%20interfere%20with%20each%20other.. [Accessed 25 04 2023].
5]
- [“Ansys,” [Online]. Available: <https://www.ansys.com/en-gb/blog/fundamentals-of-fea-meshing-for-structural-analysis#:~:text=Meshing%20is%20the%20process%20of,to%20begin%20the%20simulation%20process..> [Accessed 20 04 2023].
5]
- [“Simu Tech Group,” [Online]. Available:
6 <https://simutechgroup.com/services/consulting/structural-thermal-applications-fea/static-structural-analysis/#:~:text=A%20static%20structural%20analysis%20determines,caused%20by%20time%20varying%20loads..> [Accessed 25 04 2023].
7]
- [“Shimadzu,” [Online]. Available: https://www.shimadzu.com/an/service-support/technical-support/analysis-basics/fundament_thermal/1/index.html#:~:text=Thermal%20analysis%20is%20a%20general,substance%20is%20heated%20or%20cooled.. [Accessed 25 04 2023].
7]
- [A. Noble, “Cantilever with Uniform Load”.
8
9]
- [A. Noble, “Static Structural Study,” Andrew Noble, Swansea, 2022.
9
10]



CAE Assignment

GRADEMARK REPORT

FINAL GRADE

36/100

GENERAL COMMENTS

Instructor

You have incorporated some citations into the work in areas and there is some evidence that the research you have conducted has helped inform your discussions. However, there was scope to incorporate significantly more literature into some areas, there are statements made in several places that have no support to go with them. For example, where you discuss advantages and disadvantages of CAE, you would really want to have some literature support to go with these statements.

Areas of the discussion about the application of CAE and its possible limitations was rather brief, you could have had more extensive and justified discussion on these areas given the focus of the assignment. Some of the statements are also a bit questionable, for example you say "there are no limitations on what CAE can be used for" when there certainly are!

The description of the typical CAE process is logical and generally has the correct information in it, however there were some key areas that should have been discussed in more depth. For example, the areas of meshing and boundary conditions are possibly some of the most influential on a simulation and these were discussed very briefly. There was scope to have more depth of discussion in these areas of the report to better convey your knowledge and understanding of these factors.

You say your validation example is non-linear, but it is an elastic bending example, so still working within the expected linear range of the material under deformation. The study conducted for the validation gave results in the expected range, although you did not make this very clear from the screenshots you presented. you did not look into different mesh sizes or search for an mesh

independence point, nor did you look at boundary condition application and how influential that can be.

You finished the report with the validation study, you did not actually perform a study of your own with your own geometry which was a real shame as this was a key area of the work that was meant to be completed. The marking scheme was clear on where there were significant marks available for conducting your study with an appropriate geometry and then iterations of design based on analysis of results. This would have given you significantly more to discuss in the report when looking at the CAE process and assessing its suitability and limitations.

The report formatting was not too bad, but you wrote in the 1st person in a lot of areas, all report should be written in the 3rd person at all times. Captions were not correctly applied to figures.

The main issue here was the lack of your own study beyond just the simplistic validation study.

PAGE 1

PAGE 2

PAGE 3

PAGE 4

PAGE 5

PAGE 6

PAGE 7

PAGE 8

PAGE 9

PAGE 10

PAGE 11

PAGE 12

PAGE 13

PAGE 14

PAGE 15
