UNIVERSITY OF WALES TRINITY SAINT DAVID

SCHOOL OF ENGINEERING

ADVANCED STRESS ANALYSIS AND FINITE ELEMENT ANALYSIS

CAN FEA REPLACE PHYSICAL TESTING?

M6X00420

MAXIMILIAN HART
1700625

29-03-2020

Table of Contents

1 Introduction 3

2 Benefits of Carrying out FEA Alongside Physical testing 4

2.1 Supporting Case Studies 5

2.1.1 Impact Performance Evaluation of a Crash Cushion Design Using Finite Element Simulation and Full-Scale Crash Testing (Büyük *et al.*, 2018). 5

2.1.2 Finite Element Analysis and Experimental Characterisation of SMC Composite Car Hood Specimens under Complex Loadings (Kelly et al., 2018). 6

3 Correlating Physical Testing and FEA Model Results 7

3.1 Supporting Case Studies 7

3.1.1 Impact Performance Evaluation of a Crash Cushion Design Using Finite Element Simulation and Full-Scale Crash Testing (Büyük *et al.*, 2018). 7

3.1.2 Finite Element Analysis and Experimental Characterisation of SMC Composite Car Hood Specimens under Complex Loadings (Kelly et al., 2018). 8

4 Possibility of FEA Completely Replacing Physical Testing 10

4.1 Advantages of FEA replacing Physical Testing 10

4.2 Disadvantages of FEA replacing Physical Testing 11

4.3 Conclusion and Evaluation 11

5 Referenced Work 12

Table of Figures

Figure 1 - Example of 1D Series Spring System (Ashkan, 2006) 3
Figure 2 - FEA and Physical Models of Crash Cushion (Büyük et al., 2018) 5
Figure 3 - Physical and FEA Analysis of Composite Sample (Kelly et al., 2018) 6
Figure 4 - Visual Comparison of Simulated and Physical Tests (Büyük et al., 2018) 8
Figure 5 - Deformed Shape Comparison from Physical and FEA Tests (Büyük et al., 2018) 8
Figure 6 - Physical Testing Tensile Strain Results (Kelly et al., 2018) 9
Figure 7 - FEA Testing Tensile Strain Results (Kelly et al., 2018) 9
Figure 8 - Table of Numerical Analysis of Tensile Results (Kelly et al., 2018) 9
Figure 9 - Physical Testing Flexural Displacement Results (Kelly et al., 2018) 9
Figure 10 - FEA Testing Flexural Displacement Results (Kelly et al., 2018) 10
Figure 11 - Table of Numerical Analysis of Flexural Results (Kelly et al., 2018) 10

Table of Equations

Equation 1 - Hooke's Law 3
Equation 2 - Finding Stress from Hooke's Law (HBM, 2019) 3

1 Introduction

In almost every engineering application, stress analysis is crucial. Components within any mechanical system are likely to experience large amounts of stress, especially in a performance application. To manufacture a good product, engineering companies must fully understand these stresses to ensure chances of failure are eliminated. It is also important, especially in performance applications, to understand how to reduce the mass of a component as much as possible whilst still ensuring there is no risk of failure (Miller, 2015; Edoxx, 2018). The two most common ways of analysing the stresses acting on a component are through Finite Element Analysis (FEA) and physical testing.

FEA uses the finite element method to numerically analyse stresses acting on a component. This consists of user-selected boundary conditions that dictate a set of partial differential equations to numerically characterise the scenario (Harish, 2020). The finite element method discretises these equations through integration by parts to convert them to a set of matrix equations, allowing matrix algebra to be used to find a numerical solution of all unknowns (Morris and Rahman, 2008; Rieg *et al.*, 2014). This matrix is developed by placing a grid, known as a mesh, onto the existing geometry. This breaks down the component into smaller, more localised calculations. This makes large stress analysis more manageable and increases accuracy. Each section of the grid is known as an element, and from this each element can be analysed using Hooke's Law:

$$F = K \cdot U$$
$$Force = Spring\ Stiffness \cdot Displacement$$

Equation 1 - Hooke's Law

It is considered that the elements connect in such a way that a spring stiffness can be applied. This stiffness is quantified as $\frac{E \cdot A}{L}$, where E = Young's Modulus, A = Area and L = Length (Dean, 2012). This, and a simple springs in series arrangement, can be used to analyse the stresses of each element.

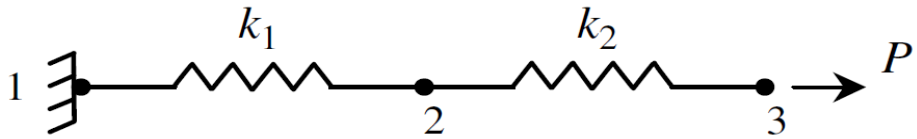

Figure 1 - Example of 1D Series Spring System (Ashkan, 2006)

Figure one shows how the nodes interact via spring rates under load from a total force 'P' exerted. This can be true for a 1D, 2D or 3D system and can be converted to matrix form for ease of calculations:

$$\begin{bmatrix} F_1 \\ F_2 \end{bmatrix} = \begin{bmatrix} K_1 & -K_1 \\ -K_1 & K_1 \end{bmatrix} \cdot \begin{bmatrix} U_1 \\ U_1 \end{bmatrix}$$

From adding the matrices of each node, an overall resolved force can be calculated, leading to localised stresses within the component by implementing:

$$\sigma = \frac{F}{A}$$
$$Stress = \frac{Force}{Area}$$

Equation 2 - Finding Stress from Hooke's Law (HBM, 2019)

Physical testing is a more traditional, experimental way of drawing the same conclusions as FEA. The stresses on a component can be measured using specialist equipment or in some cases calculated by hand from the forces applied. Strain gauges can be used to calculate the stresses in a variety of areas of a component to give an overview of the total stresses acting upon it. Even more simply, the stress at a given area of a component can be calculated from equation 2 (HBM, 2019). Other, generally less accurate techniques such as application of brittle lacquers may also be used. A brittle coating with known threshold strain can be applied to a component to test stresses applied, as the user will know what strain cracks the coating. This is a less comprehensive test but can confirm existing results (Hearn, 1997).

The equations in the later method can be taken at various points around the component and solved simultaneously through matrix algebra much like within FEA, however by hand this is very time consuming.

Both FEA and physical testing are good for calculating the physical reliability of a component and have their individual advantages and disadvantages, however using them together is always thought to be the most reliable, accurate and failsafe form of stress analysis (Dey, 2014).

This research shall critically analyse both methods, discussing how they can be used together and how using one may be more beneficial than the other in certain applications through discussion and analysis of case studies specific to the author's area of study: Automotive Engineering.

2 Benefits of Carrying out FEA Alongside Physical testing

The main disadvantage of FEA is any software is only as good as its user. There are many complex parameters that must be correctly set up by the user, any errors in which could severely impact the accuracy of results found. To ensure the FEA software has been initialised correctly and the results are accurate, it is good practice to carry out physical testing on a component alongside FEA to correlate results. It is incredibly dangerous if the computer model suggests a component is safe but the calculations were off by a factor of ten, as this could potentially mean a component would be designed to only withstand 10% of the forces it's expected to experience.

Errors within how the FEA model is set up can come from a variety of places. Some problems will require a large amount of data to be inputted by the user. Within each input, errors can be apparent. It is possible that these errors can combine in such a way that the results still look feasible, however this is down to chance and not accurate calculations. If the same setup was to be used for another example, the results may be entirely inaccurate; equally, a setup that works for one geometry will not work for all, so this inaccuracy is a threat for each model that is run (Underwood, 1989). Moreover, within the finite element method on a complex geometry, many equations need to be simultaneously solved to come to a solution. Within each of these equations, a rounding error can cause inaccuracy; even the best computers around can only work with a certain number of decimal places, and although this may only cause a very small error, with so many equations, when they add up it could result in a large overall error in the final result, known as a truncation error (Langtangen, 2014).

Because of this, it is always a good idea to correlate FEA results with real life results. If the results of both are directly comparable, it is clear that the numbers from FEA are trustworthy. For scenarios surrounding one geometry it is acceptable to suggest that only two physical and FEA tests need to be correlated, which is enough to confirm that the FEA model being used is producing accurate results (Wen, 2007). Once this confirmation is in place, the user may confidently continue using FEA to analyse multiple design iterations without the time constraints of producing and testing physical models. This really is the main benefit of FEA. Once confirmation that the model is producing accurate results has been obtained, the user can save huge amounts of time by eliminating time spent producing and testing physical models (Everett, 1994).

2.1 Supporting Case Studies

2.1.1 Impact Performance Evaluation of a Crash Cushion Design Using Finite Element Simulation and Full-Scale Crash Testing (Büyük *et al.*, 2018).

An impact attenuator, otherwise known as a crash cushion, is designed to gradually absorb the kinetic energy of a collision between it and a vehicle. There is a set maximum deceleration that the vehicle must not exceed to ensure the safety of the occupants. The idea is for the crash cushion to be entirely sacrificial, limiting vehicle damage as much as possible (Trinity Highway, 2019).

Due to the sacrificial nature of this product, and the fact that staging a car crash is the only accurate way of testing it, product development tests are incredibly expensive. For this reason, FEA is a fantastic way of validating physical testing. It is considerably cheaper to find an existing computer package and experienced user to run tests than to stage real-life tests. Once the FEA is completed, the results can be analysed to ensure the crash cushion is acting as expected. If the results look good, the user can be confident that staging a real-life test will not be a waste of money (Grabianowski, 2017). The real-life test then acts as validation of the FEA, which can be compared to ensure neither results are erroneous.

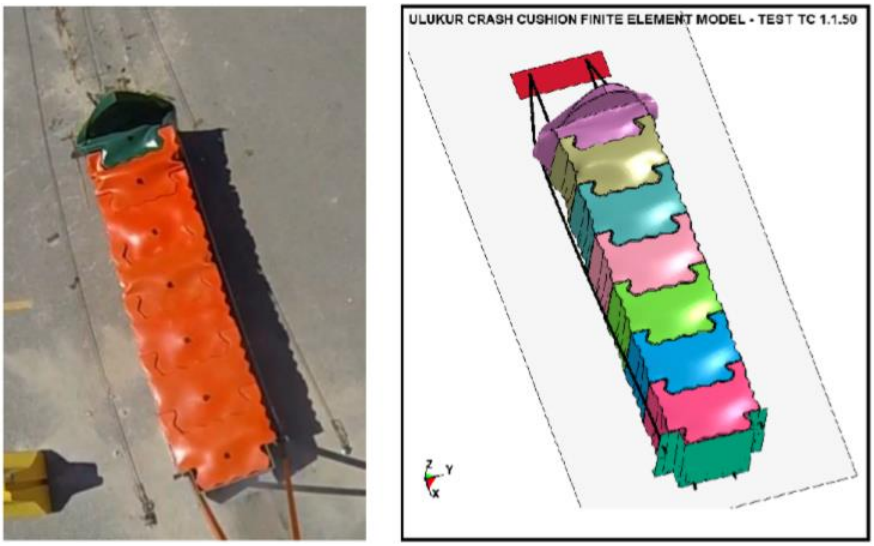

Figure 2 - FEA and Physical Models of Crash Cushion (Büyük et al., 2018)

2.1.2 Finite Element Analysis and Experimental Characterisation of SMC Composite Car Hood Specimens under Complex Loadings (Kelly et al., 2018).

Composite materials are quickly becoming more popular in the automotive industry due to their light weight and high strength. However, their behaviour at failure is very different to most metals and is widely thought to be understood by FEA applications less than metals (Adams, 2012). To understand the advantages of use of composites within a car hood [bonnet], samples of the materials were taken for this report and stress tested.

The report hypothesised that the findings of the FEA would become less accurate at higher applied forces, producing a result of higher volume fraction percentages. The report states that this would encourage a higher chance of fibre-related failure modes such as kinking or rupture. These faults within the model cause localised failure, causing the part to fail before reaching ultimate tensile strain, creating less trustworthy results. Moreover, the report explains that the FEA model used is based on a tensile finite model, accounting for the tensile properties of the material, but not the compression; this, of course, is a further source of inaccuracy.

The hypothesised inaccuracies were apparent in the FEA results; however they were close to what was expected and within tolerance, showing validity (Pederson, 2008). This showed it was worth investing in physical testing. Taking the results from both tests, the inaccuracy within the FEA model was properly understood, but the correlation of results validated both. By understanding the percentage errors of the FEA, the user may not have to conduct physical testing in future work, thus saving time and money.

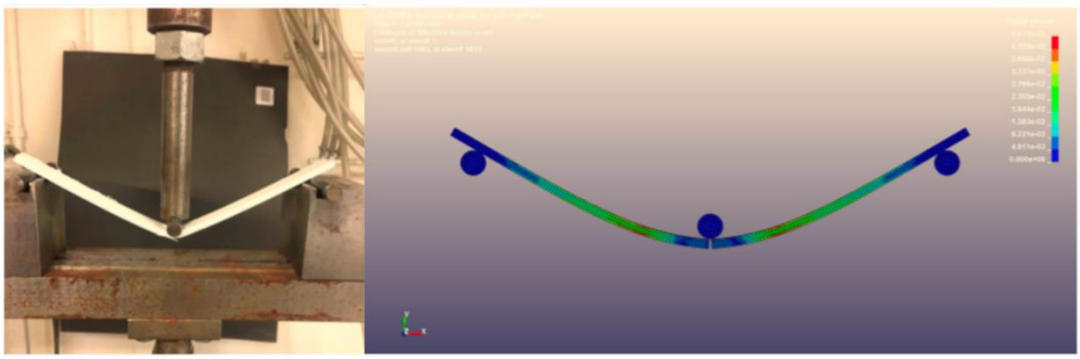

Figure 3 - Physical and FEA Analysis of Composite Sample (Kelly et al., 2018)

3 Correlating Physical Testing and FEA Model Results

For every stress analysis test conducted, a specific result will be important to the user. This may be the specific stress such as tensile, bending or Von Mises stresses, deformation or resultant deceleration acting on a body in a certain scenario. FEA is designed to replicate real-world, stress imposing situations to produce results. Because of this, a real-world test can always theoretically be conducted to achieve the same results, and be directly comparable (Hemez and Doebling, 1998; Behrens *et al.*, 2014).

To correlate physical testing as accurately as possible, a reference solution must first be found. This can be either experimental or computational, however it is usually the latter as this is generally a cheaper, faster way of finding initial results. From this, baseline results can be found to be replicated by other means later on (Bathe, 2014). Correlation will depend on the type of study being carried out, as generally FEA will produce very accurate results with simple, linear elastic models; however, as the loading scenario and resultant stresses become more complex and non-linear, the finite element method becomes more inaccurate (Bathe, 2014). Every FEA test should be carried out more than once to ensure no accidental mistakes were made when setting up the simulation, and ideally physical testing should be done more than once also to eliminate the chance of random error, potentially from equipment, measurement or user error (Bi, 2018).

As previously mentioned, the user may use strain gauges or overall hand calculations to recreate FEA results. It is a good approach to physically test the worst-case scenario on the component. For example within an engine connecting rod study, the user may run FEA simulation on stresses experienced at every RPM; however, only physically test the component at the forces seen at the RPM producing the highest cylinder pressure. This way, the user can analyse the effects on the component when most likely to fail. If this crucial point correlates with the FEA results, they can be deemed trustworthy (Bathe, 2014).

3.1 Supporting Case Studies

3.1.1 Impact Performance Evaluation of a Crash Cushion Design Using Finite Element Simulation and Full-Scale Crash Testing (Büyük *et al.*, 2018).

This report focused primarily on the deceleration of the vehicle upon impact with the crash cushion. This is something the chosen FEA software (LS-DYNA, a multiphysics simulation software package specialising in impact deformation) can accurately calculate (Livermore Software Technology, 2015). The user was also interested in how the crash cushion deformed compared to the vehicle. Upon completing both tests, the deformation was both numerically and visually compared as well as confirming the deceleration was below the minimum safe threshold.

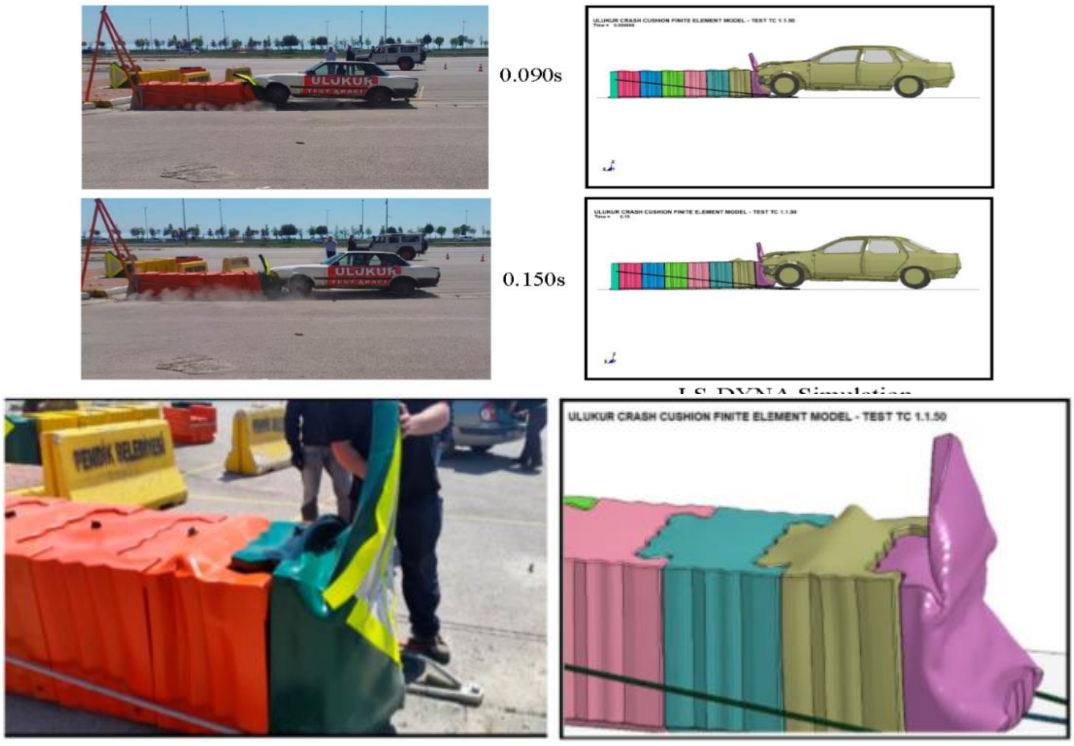

Figure 5 - Deformed Shape Comparison from Physical and FEA Tests (Büyük et al., 2018)

It is clear visually that the FEA analysis accurately predicted the severe deformation of the first module of the crash cushion, and mild deformation of the second. It is also clear that the vehicle was only lightly damaged, with no damage to the passenger occupancy area. Accelerometers within the real car confirmed the FEA results of deceleration to be safe. This confirms many things; the design works well and protects the occupants of the vehicle, and the design also is cost-effective as only one or two modules need to be replaced, reducing the cost of replacing an entire barrier (Mori *et al.*, 1983; Trinity Highway, 2019).

3.1.2 Finite Element Analysis and Experimental Characterisation of SMC Composite Car Hood Specimens under Complex Loadings (Kelly et al., 2018).

From the material sample tested, the tensile strain and flexural displacement were measured in both longitudinal and transverse directions. The data was taken directly from the FEA results and calculated from the results of the physical testing so a comparison could be made.

3.1.2.1 Tensile Strain

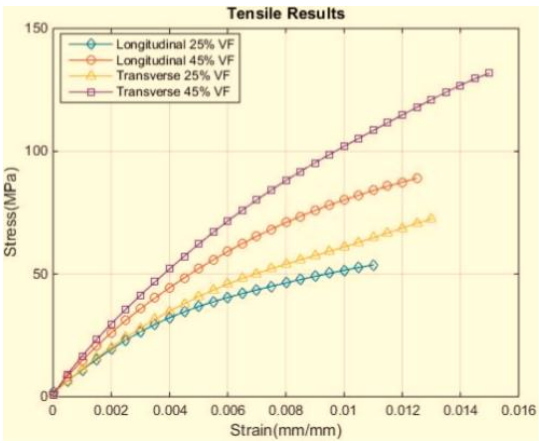

Figure 6 - Physical Testing Tensile Strain Results (Kelly et al., 2018)

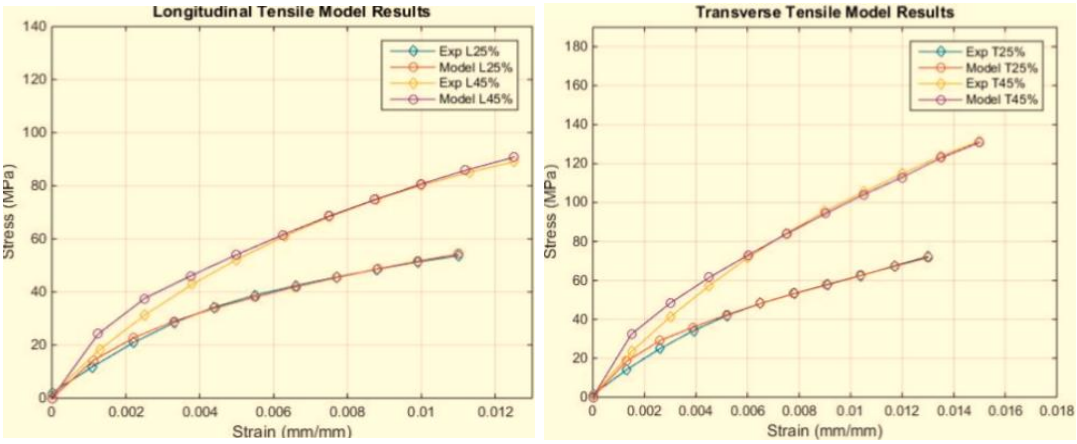

Figure 7 - FEA Testing Tensile Strain Results (Kelly et al., 2018)

Volume Fraction	UTS (MPa) Experiential	UTS (MPa) Simulation	% Difference	$\varepsilon_{Fracture}$ Experimental	$\varepsilon_{Fracture}$ Simulation	% Difference	RMS Error (MPa)
L25%	53.7	54.3	1.10%	1.10%	1.10%	0%	0.99
L45%	89	90.7	1.90%	1.25%	1.25%	0%	2.98
T25%	72.4	71.7	1.00%	1.30%	1.30%	0%	1.97
T45%	131.7	131	0.50%	1.50%	1.50%	0%	4

Figure 8 - Table of Numerical Analysis of Tensile Results (Kelly et al., 2018)

Figures six to eight clearly show how data is correlated from the FEA and physical tests. Each result is graphed to visually compare trends then tabulated with percentage differences and errors shown. This clearly displays the errors mentioned in section 2.1.2 and shows the expected trend of larger error at higher forces applied. As previously mentioned, this data was then used within the report to predict FEA error at higher applied force, eliminating the need for physical testing in further iterations.

3.1.2.2 Flexural Displacement

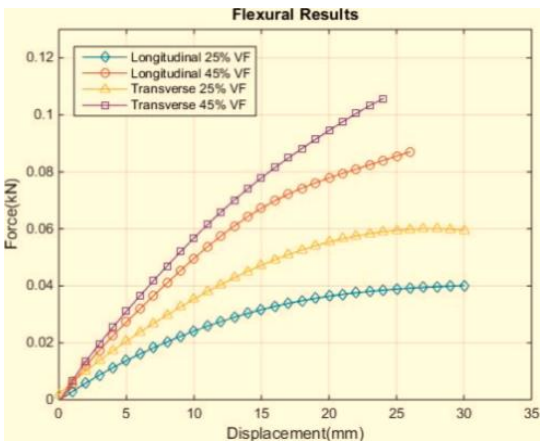

Figure 9 - Physical Testing Flexural Displacement Results (Kelly et al., 2018)

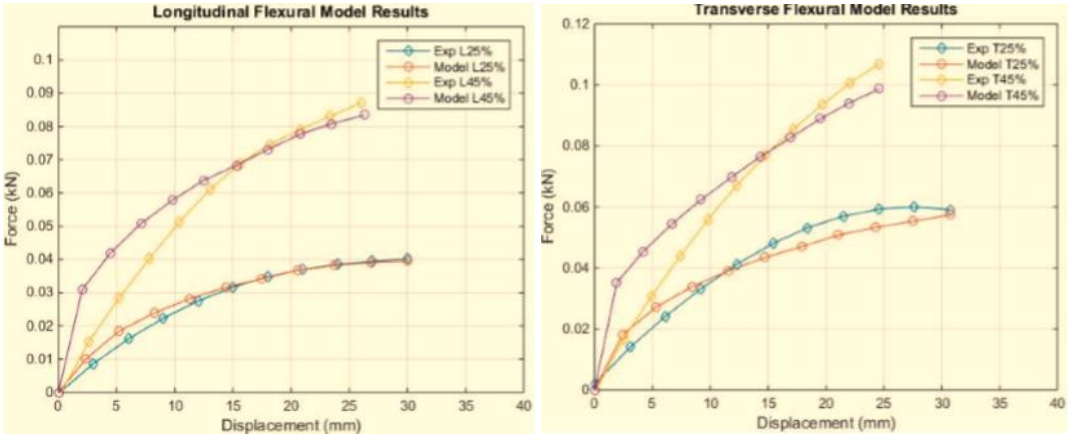

Figure 10 - FEA Testing Flexural Displacement Results (Kelly et al., 2018)

Volume Fraction	Failure Force (N) Experiential	Failure Force (N) Simulation	% Difference	Displacement (mm) Experimental	Displacement (mm) Simulation	% Difference	RMS Error (N)
L25%	40.1	39.5	1.50%	30	30	0%	1.53
L45%	87.1	83.4	4.20%	26	26.3	1.20%	6.22
T25%	59.5	57.4	3.50%	30	30.7	2.30%	3.39
T45%	105.7	98.7	6.60%	24	24.6	2.50%	6.74

Figure 11 - Table of Numerical Analysis of Flexural Results (Kelly et al., 2018)

Figures nine to eleven also clearly show expected trends and allow the user to form good ideas of how FEA results can be altered to be more accurate at higher applied forces. From these results, factors of error for this setup can be found and applied to future iterations to get a more representative result. These trends support the previously mentioned advantages to FEA modelling; once a small number of physical tests have been correlated to the simulated results, there is no need to run any further physical testing, saving both time and money (Pederson, 2008).

Sections 3.1.1 and 3.1.2 demonstrate how physical and FEA testing can be correlated through both visual and numerical analysis; both confirmed the trends that the authors expected, while giving further confidence in the results through repetition of tests.

4 Possibility of FEA Completely Replacing Physical Testing

4.1 Advantages of FEA replacing Physical Testing

As discussed throughout this research and confirmed by analysis of case studies, there are many clear advantages to using FEA. It is widely regarded as one of the fastest growing disciplines of engineering and is becoming increasingly more popular, this would not be the case if it was not a valuable tool.

FEA can give the user a comprehensive overview of the stresses acting on a component, providing the user selects an appropriate, accurate mesh. This is beneficial as some areas may be missed out in physical testing as only certain areas chosen by the user to place gauges and sensors on or perform calculations on are analysed (Pre Technologies, 2020). By this logic, FEA reduces the chance of an engineer overlooking a critical failure point within a component. Equally, FEA offers a visual representation of a stress system. This gives a huge advantage over physical testing as areas of weakening may not be apparent in real-world applications. Some materials can be severely damaged and show little sign of stress. This, again reduces the chance of overlooking the possibility of failure (Miller, 2015).

FEA also offers the user a safe way to test potentially dangerous scenarios. If a component needs to be tested to destruction, physical testing could have catastrophic consequences. For example, a wishbone within a car's suspension system; to test this to destruction a vehicle must hit a large bump at high speed, which is not a safe or easy test to run (Hearn, 1997). It could be argued that the most important advantage of FEA is its ability to save time and money. This has been demonstrated many times within this research and is the fundamental advantage to computational stress analysis. If tested to destruction, the model can simply be reset and the testing can start again. There is no need for rebuilding multiple models for multiple iterations; this can save vast amounts of money throughout a component's research and development stages. Moreover, not having to build multiple models saves time, along with computational power having the ability to solve complex matrix algebra in a fraction of the time it would take an engineer (Tovey, 2002).

4.2 Disadvantages of FEA replacing Physical Testing

As discussed throughout this research, there are many disadvantages to FEA testing. Most notably, inaccuracy within results. FEA accuracy is down to the user. If a model is poorly set up, results found will be invalid (Bathe, 2014).

For anything more than simple geometries, large amounts of data are required to be inputted to the model and the software has to perform large amounts of calculations on them. Most component geometry will be of a complex nature, with a sophisticated, fine mesh needed to produce any valid results. This increases the likelihood of errors within the model and can increase the time taken to reach a result. Even with a good user setup, FEA packages are generally accepted to be very accurate at modelling component stresses within the region of linear deformation, however beyond the yield point, in the plastic region, results can be less trustworthy. This is generally acceptable as most component testing is analysing the behaviour up until the point of deformation; however it is important to understand behaviour after this point also, and to confirm results physical testing is often required (Dean, 2012; Javidinejad, 2012).

It is important to use the correct FEA package for the test being carried out. There are many different options available and each vary in levels of accuracy. If an engineer is in a situation where the best package is unavailable due to lack of resources or funding, their results may be impaired. (VGO Inc., 2020). Moreover, if results are in the public eye, physical testing is more understandable. For example, Euro NCAP safety tests are predominantly carried out in real-world scenarios as the consumer can physically see and trust results demonstrating the safety of the product they are buying, FEA results are outside most people's understanding, which could lead to them being perceived as untrustworthy, which could affect sales.

4.3 Conclusion and Evaluation

Overall, due to the nature of FEA and the large margin for user error, the author does not foresee physical testing ever being completely replaced, at very least not in the near future. Physical testing will still be needed to correlate simulated findings. It is clear that the best compromise is to carry out a limited number of initial physical tests to correlate to initial FEA tests; once correlation is found showing valid results, FEA tests can be run on more iterations of the component to save time and money. Due to the high risk of error, the author does not believe FEA models alone can always be taken as completely trustworthy.

5 Referenced Work

Adams, D. (2012) *Composites in Cars: Making Vehicles Lighter, Safer and More Fuel-Efficient*. Utah. Available at: https://mech.utah.edu/composites_cars/.

Ashkan, F. (2006) *1D Spring Element FE code*. Available at: https://uk.mathworks.com/matlabcentral/fileexchange/45155-1d-spring-element-fe-code?focused=3806420&tab=function&s_tid=gn_loc_drop.

Bathe, K.-J. (2014) *Finite Element Procedures*. 2nd edn. Massachusetts: Pearson. Available at: http://web.mit.edu/kjb/www/Books/FEP_2nd_Edition_4th_Printing.pdf.

Behrens, B. A. *et al.* (2014) *Material Modelling of Short Fiber Reinforced Thermoplastic for the FEA of a Clinching Test*. Hannover. Available at: https://www.sciencedirect.com/science/article/pii/S2212827114006283.

Bi, Z. (2018) *Finite Element Analysis Applications : A systematic and Practical Approach*. 1st edn. London: Elsevier.

Büyük, M., Atahan, A. O. and Kurucuoğlu, K. (2018) *Impact performance evaluation of a crash cushion design using finite element simulation and full-scale crash testing*, *Safety*. doi: 10.3390/safety4040048.

Dean, J. (2012) *Introduction to the Finite Element Method (FEM)*, *Lecture Notes*. Cambridge. Available at: https://www.ccg.msm.cam.ac.uk/images/FEMOR_Lecture_1.pdf.

Dey, A. K. (2014) *Basics of Pipe Stress Analysis – FEA and Physical Testing*. Available at: https://whatispiping.com/basics-of-pipe-stress-analysis.

Edoxx (2018) *Why It ' s Important To Perform A Stress Analysis*. Available at: https://www.edoxx.com/single-post/2017/05/19/Why-Its-Important-To-Perform-A-Stress-Analysis.

Everett, A. (1994) *Materials*. Harlow: Longman Group Ltd.

Grabianowski, E. (2017) *How Crumple Zones Work*. Available at: https://auto.howstuffworks.com/car-driving-safety/safety-regulatory-devices/crumple-zone1.htm.

Harish, A. (2020) *What Is FEM and FEA - Explained*. Available at: https://www.simscale.com/blog/2016/10/what-is-finite-element-method/.

HBM (2019) *Experimental Stress Analysis (ESA) using Strain Gauges | HBM*. Available at: https://www.hbm.com/en/7226/experimental-stress-analysis-using-strain-gauges/.

Hearn, E. J. (1997) *Experimental Stress Analysis - an overview*. Available at: https://www.sciencedirect.com/topics/materials-science/experimental-stress-analysis.

Hemez, F. M. and Doebling, S. W. (1998) *Test-Analysis Correlation and Finite Element Model Updating for Nonlinear, Transient Dynamics*.

Javidinejad, A. (2012) *FEA Practical Illustration of Mesh-Quality-Results Differences between Structured Mesh and Unstructured Mesh*, *ISRN Mechanical Engineering*. doi: 10.5402/2012/168941.

Kelly, J., Cyr, E. and Mohammadi, M. (2018) *Finite Element Analysis and Experimental Characterisation of SMC Composite Car Hood Specimens under Complex Loadings*, *Journal of Composites Science*. doi: 10.3390/jcs2030053.

Langtangen, H. P. (2014) *Truncation Error Analysis*. Available at: http://hplgit.github.io/INF5620/doc/pub/H14/trunc/html/._main_trunc000.html (Accessed: 23 December 2019).

Livermore Software Technology (2015) *LS-DYNA | Livermore Software Technology Corp.* Available at: http://www.lstc.com/products/ls-dyna (Accessed: 25 March 2020).

Miller, K. (2015) *Measuring Material Properties to Build Material Models in FEA*. Available at: https://dspace.lib.cranfield.ac.uk/handle/1826/10425.

Mori, K., Osakada, K. and Fukuda, M. (1983) 'Simulation of severe plastic deformation by finite element method with spatially fixed elements', *International Journal of Mechanical Sciences*. Pergamon, 25(11). doi: 10.1016/0020-7403(83)90060-7.

Morris, A. and Rahman, A. (2008) *A Practical Guide to Reliable Finite Element Modelling*, *A Practical Guide to Reliable Finite Element Modelling*. John Wiley and Sons. doi: 10.1002/9780470512111.

Pederson, J. (2008) *Finite Element Analysis of Carbon Fiber Composite Ripping Using ABAQUS*. Available at: https://tigerprints.clemson.edu/all_theses.

Pre Technologies (2020) *Advantages of FEA*. Available at: http://www.pretechnologies.com/services/finite-element-analysis/advantages.

Rieg, F., Be, R. H. and Alber-Laukant, N. (2014) *Finite Element Analysis for Engineers*. Cincinnati: Hanser Publications. Available at: https://play.google.com/store/apps/details?id=z88tina.fr.

Tovey, M. (2002) 'Concept design CAD for the automotive industry', *Journal of Engineering Design*, 13(1), pp. 5–18. doi: 10.1080/09544820110090287.

Trinity Highway (2019) *REACT 350 Product Specificaton - Trinity Highway*. Available at: https://trinityhighway.com/product/react-350/.

Underwood, J. H. (1989) *Applications and Limitations of Finite Element Analysis To Armament Components*, *Technical Report ARCCB-TR- 89018*. New York.

VGO Inc. (2020) *Dangers of Modern FEA*. Available at: http://vgoinc.com/general/dangers-of-modern-fea.

Wen, J. H. (2007) 'FEA modeling verification and validation: Correlating model with test data by optimization analysis', in *SAE Technical Papers*. SAE International. doi: 10.4271/2007-01-1745.