
Mohamad Saleh
School of Informatics and Engineering, Institute of Technology, Blanchardstown, Dublin 15, mohamad.saleh@itb.ie

Follow this and additional works at: https://arrow.tudublin.ie/itbj
Part of the Numerical Analysis and Scientific Computing Commons

Recommended Citation
Available at: https://arrow.tudublin.ie/itbj/vol5/iss1/23

This work is licensed under a Creative Commons Attribution-Noncommercial-Share Alike 3.0 License

Mohamad Saleh B.Sc. M.Eng., Ph.D., CEng, MIEE

School of Informatics and Engineering, Institute of Technology, Blanchardstown, Dublin 15
Contact email: mohamad.saleh@itb.ie

Abstract

Mechatronics design process is a series of analytical brainstorming operations from specification to implementation. The mechatronic products are widely available in the market for various use and applications. This is expected to develop further in the future with great competitiveness. In order to succeed in the market, mechatronic products need to satisfy the market expectations with regard to quality, fitness for purpose, customer’s appeal and cost efficiency. Therefore, the design analysis techniques play a significant part in the market success of these products.

Finite Element Analysis is probably the most commonly used numerical technique for mechatronic product design. This technique has been developed over a number of years and is now available in a wide variety of packages, mostly for mainframe and workstation platforms but also for personal computer systems. Over the past few years, the range of users has broadened dramatically with a rapid need for training and education of this technique.

This paper reviews the design philosophy and modelling strategy of a mechatronic product using finite element techniques. This takes into account the design study of a 140 tonne load cell for measuring a load mechanism in metal working applications.

Keywords: FEA, transducer, Mechatronic, product design.

Introduction

Mechatronics is the synergic combination of precision mechanical engineering, electronic control and systems thinking in the design of products and processes (Reitdijk, J. A). It is therefore important that mechatronics, from the very beginning, is considered not as a separate engineering discipline but instead as an integrating systems level approach to design and operation of a wide range of complex engineering productions and processes (D.A Bradley, 1997). Thus, mechatronics, at undergraduate level, should be delivered as a compatible method to its integrated and inter-disciplinary nature. M. Saleh, 2003, discussed an effective approach in mechatronics design and its implementation at third level mechatronics courses.

Finite element method has become the most reliably effective numerical technique used to calculate the most complicated engineering problems compared to other computer aided engineering. The popularity of this technique is due to its wide applicability in both static or dynamic linear and non-linear problems, In fact, continuous fast advancement in computer hardware and software and the availability of computer power have caused a dramatic increase in the use of finite element techniques in engineering design. This has led to the belief that the development of educational and training courses in finite element techniques is crucially
important to extend the popularity of this technique for obtaining quality and reliable results of the modern product design.

The material presented in this paper is a revised and expanded version of a lead paper presented by M. Saleh, 1999, at AMPT’99 [4]. This paper discusses a practical approach for using finite element techniques for reliable design and analysis. It takes into account the systematic scenario and strategy of FEA modelling which is enhanced through a practical example. It is believed that the implementation of this approach in training and educational programmes will benefit the designers, as it improves the productivity, reduces analytical costs and hence more accountable quality design can be seen.

1. Finite element technique

Finite element is a method of mathematically modelling a components/structure for stress analysis. It requires large quantities of data which are manipulated by matrix techniques using the appropriate computer software programmes. It is not intended to give a detailed account of finite element techniques, as this is well documented in numerous text books. However, it is felt that a general overview will enhance the understanding of the subsequent work reported in this paper. Figure 1 shows the procedure of finite element technique to solve the design parameters of a cantilever loaded with a point load at its free end. This problem can be solved using two or three dimensional F.E modelling analysis. In three dimensional analysis the entire structure of this cantilever will be modelled using three dimensional elements. Often in stress analysis a full three dimensional treatment of a problem can be avoided by assuming that the structure to be adequately represented if plain stress or plane strain conditions are adapted. It is assumed that plane strain conditions are justified for the cantilever in Figure 1.a when the ratio b/h exceed 5. This means that there is no deformation in Z direction. However plane stress conditions are justified when b/h is less than 5.

Following on from this, the structure of the cantilever will be divided into a mesh of elements, as shown in Figure 1.b with the appropriate material properties, boundary conditions at the restraints and external loading types. In this mesh, each node of the individual element has two degrees of freedom (Figure. 1 .c) making a total of eight nodal displacements for the element. Similarly, each element in this mesh has eight nodal forces. The concept of element stiffness matrix [K'] is introduced by relating the nodal forces to the nodal displacements of the element Figure 1 .d. The overall stiffness matrix [K] in Figure i.e can be established by assembling the element stiffness matrices of the overall structures. From the equation in Figure 1.e, the unknown displacements for each individual node in the mesh can be found. Thus, the stress and
strain for each element in the structure can be calculated using the formulas shown in Figure 1.f.

This sequence of operations can be performed using the interactive finite element commercial software programmes. These programmes, in general, feature the same philosophy with different sizes of analysis and slight variations of menu driven functions and operations. The accuracy and reliability of results depend upon the finite element modelling strategy in which choosing the appropriate element, material properties, boundary conditions and applied load must be defined precisely. In practice, the finite element solution of an engineering problem may be varied among different users using the same FE programme. Also, errors in finite element solutions can be produced by the F.E programmes. Consequently, agencies such as NAFMES (National Agency for Finite Methods and Standards) in England play a most important role with their regular survey report of tests carried out on leading finite element software programmes worldwide. These are reported in NAFEMS periodical “Benchmarks”.

The unfamiliarity with finite element methods and relevant software programmes produces erroneous answers to problems using finite element modeling. Recently, this has led to increased demand for training and education in finite element modelling techniques. This is to enable the trainees to carry out a reliable finite element analysis, under critical working conditions, of their engineering design.

\[ \{ \varepsilon^e \} = [B] \{ \varepsilon^e \} \]

\[ \{ \sigma^e \} = [D] \{ \varepsilon^e \} \]

Figure 1: Procedure of finite element analysis

2. Modelling strategy

In order to enable the structure to maintain its characteristic safety in critical situations, the finite element model should be approached with a sound basic knowledge of the finite element
technique and its applications to engineering problems. Therefore, the users must have an appreciation for discretisation principles, element performance, modelling procedures and material constitutive behaviour, as the reliability of any finite element solution depends fundamentally on the consideration of these factors and the background to the field of application.

Figure 2 shows the sequence of a systematic procedure for using finite element technique as a reliable computer-aided design and analysis in engineering. This procedure takes into account the theoretical model, experimental model and the design goal of the problem being investigated. The theoretical model considers the finite element and the conventional analytical model of the problem. The conventional analytical model is based on the traditional calculations of stress and strain in engineering design. This is to determine the design specifications, to develop the experimental model and consequently to validate the refinement of the finite element model. The finite element model is a scenario of model modifications with different types of elements, boundary conditions, material properties, mesh refinement; and theoretical design modifications. The design goal is the permissible design parameters/specifications taken from international literature or standard systems. The experimental model involves the physical development of the design in question, including the appropriate measurement system. These are to make comparative studies and to validate the finite element solutions.

Having introduced the first finite element model, comparison takes place between this model, experimental model and the conventional analytical model. This is to establish a referenced correlation between these models. The negative decision at the end of this comparison leads to more modifications/refinements of the finite element model. The positive decision means that the finite element model is valid and the theoretical modifications of the design can be made. This involves a series of theoretical changes of the design and relevant finite element modelling. The objective of this is to optimise the design functions with respect to the design goal. In this regard, the negative comparison suggests that more theoretical modifications of the design should be made. However, the positive comparison leads to applying the theoretical modifications on the actual experimental model. Following on from this, comparison will be carried out to validate the final design solution. Consequently, a new design and analysis will be introduced with practical conclusion.
3. Design philosophy

In order to enhance the understanding of the modelling strategy presented in this paper, the development of the F.E model of a load cell of 140 tones, shown in Figure 3, is described. The principle of four bridge single strain gauges is used in the design to translate the physical applied load to an electrical signal. This electrical signal can be sent to a data acquisition system based computer to further evaluation and accurate read out of the applied load. The philosophy of the design was to optimise the structural design functions of this cell so that it can perform safely under the critical loading conditions. The non-linearity portion of the output signal was considered and consequently a clamping system with the relevant accessories was implemented in the design to overcome the non-linear loading during the calibration process.

4. Finite element model

The finite element model took advantage of the axisymmetrical geometry of the load cell to reduce the time and the cost of the finite element analysis. LUSAS 10.1 FE software package was used as a computer aided design and analysis. Accordingly, an axisymmetric finite element model of the load cell was introduced as shown in Figure 4. This employed axisymmetric elements with 2d.f7node and static faced load was applied externally. The mating elements of the load cell were modelled using the non-linear boundary conditions and the model was restrained as follows:
(i) Nodes on centre line were restrained in X direction.
(ii) Nodes on B1-B2 were restrained Y direction.

![Diagram of Load Cell](image)

**Figure 3: Load cell**

Material properties were considered for tool steel (AISI D2). The design goal was counted as the permissible stress for this steel. The model underwent a scenario of theoretical modifications to optimise the dimensions; to give as constant stress as possible across the thickness (t) at half of the height (L) and to minimise the concentrated stresses at radius r and r3. The final results of the stress distribution and deformation mode are shown in Figure 4.b and Figure 4.c respectively.

![FEA Model and Results](image)

**Figure 4: FEA Model and results**
5. Calibration procedure

This procedure is to establish the relationship between the external applied load and the output of the load cell, and thus the sensitivity of the load cell. The calibration procedure was carried out on a 250 KN Instron machine as shown in in Figure 5. A 100 KN was considered as pre-load to overcome the non-linearity encountered during the process of calibration. Thus the sensitivity of the load cell is found as follows:

\[
\text{Sensitivity} = 1.95 \mu V/KN/V_i
\]

(1)

This mean that the sensitivity of the load cell is 1.9 \( \mu \)V for each KN per excitation voltages. Assuming a 6 VDC excitation voltage \( V_i \) with an amplification 200 and 100KN external applied load, then the output voltage of the load cell is:

\[
\begin{align*}
V_o &= 1.95 \mu V \times 100 \times 6 \times 200 \\
V_o &= 0.234 \text{ Volts}
\end{align*}
\]

(2)

From equation 1 and 2, the applied load is KN is :

\[
\text{Load (KN)} = \frac{V_o (\text{Volts})}{1.95 \times 10^6 \times 200 \times 6}
\]

(3)

The calibration curve of the load cell is shown in Figure 6

![Figure 5: Calibration procedure](image-url)
6. Conclusion

As it can be seen a practical approach for using finite element for engineering design and analysis has been described. This was supported with a FEA modelling strategy of a functional 140 tonnes load cell as an example of mechatronic products.

It is believed that the material presented in this paper could be of great benefit for the designers who use finite element techniques as a design tool. In addition, this can be implemented and adapted in training and educational courses in third level or industrial establishments to broaden the effectiveness and the popularity of FEA among the engineering community.

7. References