Modelling optimization involving different types of elements in finite element analysis

2013 IOP Conf. Ser.: Mater. Sci. Eng. 50 012036

(http://iopscience.iop.org/1757-899X/50/1/012036)

Modelling optimization involving different types of elements in finite element analysis

C M Wai[a], Ahmad Rivai and Omar Bapokutty
Faculty of Mechanical Engineering
Universiti Teknikal Malaysia Melaka (UTeM)
Hang Tuah Jaya 76100, Melaka, Malaysia

E-mail: [a]cmwai@live.com.my

Abstract. Finite elements are used to express the mechanical behaviour of a structure in finite element analysis. Therefore, the selection of the elements determines the quality of the analysis. The aim of this paper is to compare and contrast 1D element, 2D element, and 3D element used in finite element analysis. A simple case study was carried out on a standard W460x74 I-beam. The I-beam was modelled and analyzed statically with 1D elements, 2D elements and 3D elements. The results for the three separate finite element models were compared in terms of stresses, deformation and displacement of the I-beam. All three finite element models yield satisfactory results with acceptable errors. The advantages and limitations of these elements are discussed. 1D elements offer simplicity although lacking in their ability to model complicated geometry. 2D elements and 3D elements provide more detail yet sophisticated results which require more time and computer memory in the modelling process. It is also found that the choice of element in finite element analysis is influence by a few factors such as the geometry of the structure, desired analysis results, and the capability of the computer.

1. Introduction

Finite Element Analysis (FEA), which was first developed for the purpose of aerospace structural analysis, is now widely used in other industries, including automotive [1-3], structural engineering [4-7], composite design and manufacturing [8-10], as a convenient and speedy tool for approximation of the solution to a variety of complicated engineering problems. FEA is the practical application of the finite element method (FEM), which is basically an alternative method to solve engineering problems numerically. Instead of using ordinary differential equation and partial differential equation, FEM divides the body into small but finite pieces called elements. Equations are formulated for each element and the results are combined to obtain the final solution to the problem [11-12]. Subdivision of the body into suitable elements is therefore a crucial step in FEA.

There are many different types of elements used in FEA. These elements are developed independently and vary from one finite element (FE) software to another. In general, there are three groups of element which are one-dimensional (1D), two-dimensional (2D), and three-dimensional (3D) elements. Logically, all virtual structures should be modelled as close as possible to the actual structure. However, 3D solid elements are usually not the best choice for FEA. The analysts should learn when other element types can be used and valid in representing the engineering structure.

Since the discussion and also the analytical model to justify the effect of elements in FE modelling is lacking, this paper aims to compare and contrast 1D, 2D, and 3D elements by literature while the advantages and limitations of 1D, 2D and 3D elements are investigated by carrying out a FEA of a simple case study of an I-beam. This paper will provide fundamental information relative to some of the finite elements used in modelling by commercial FE software today. However, it cannot include all the different types of finite elements, which is immense and so specialized that they are beyond the limits of this study.

2. Types of Elements

Elements in FEA are generally grouped into 1D element, 2D element, and 3D element. They are recognised based on their shapes. For example, elements can take on the form of a straight line or curve, triangle or quadrilateral, tetrahedral and many more. The simplest element is a line made of two nodes. All line elements, whether straight or curved, are called 1D element in which they have translational and rotational displacement functions [13]. Examples of 1D element are truss element and beam element [14].

2D elements are typically surface elements with triangle or quadrilateral as their basic shapes [13]. Examples of 2D elements are 3-node triangular element, 6-node triangular element, and many more [14]. These surface elements can have regular or irregular shapes shown in figure 1. 2D elements are plane elements. Therefore, linear approximation of translational displacements considered are u(x,y) and v(x,y) while rotational displacements are θ(x,y) [12,15]. Since they account for plane stress and plain strain, they are often used to solve 2D elasticity problems [13-14].

3D elements are usually used to mesh volumes. They are derived from 2D elements are used when the problem is unable to be simplified [16]. 3D solid elements only accounts for translational displacements. The three translational unknown displacement functions are u(x,y,z), v(x,y,z), and w(x,y,z) [15-16]. Examples of 3D solid elements are 4-node tetrahedral element, 10-node tetrahedral element, 8-node isoparametric element, etc [14].

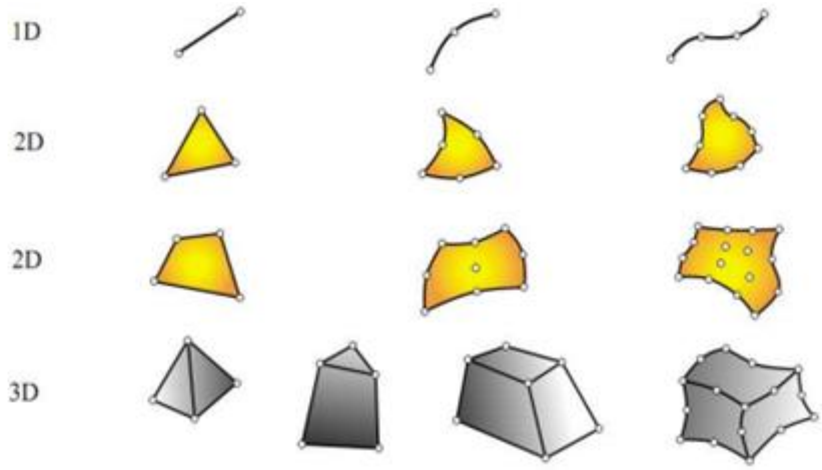


Figure 1. Typical finite element geometries [16].

3. Modelling

A wide flange beam, W460x74, is subjected to an eccentric load of 1000 kN as shown in the figure 2 below. Since the load is applied 200 mm from the neutral axis of the beam, a bending moment is induced at the end of the beam. The other end of the beam is assumed to be fixed with no translational or rotational displacements. The cross section of the beam, which was taken from Beer et. Al. [17], is shown in figure 2. The beam is assumed to be made of steel. Table 1 contains the necessary geometric dimensions and parameters of the beam. The beam is modelled and analyzed with 1D element, 2D element, and 3D element using MSC Patran and MD Nastran. In order to compare the memory and execution time of the solver, the finite elements used are of the same global edge length which is fixed at 100 mm. In this simple case study, the eccentric load applied on the I-beam induced a normal stress in the x-axis and a bending stress which causes the beam to bend upwards. In order to model the beam with 1D element, the load is decomposed into an equivalent force and moment at the centroid of the beam x-axis. The same loading condition is applied to models of 2D and 3D elements to eliminate the effect of loading conditions on the choice of element. Only static analysis which yields the results for stress, displacement, and constraint forces is taken into consideration.

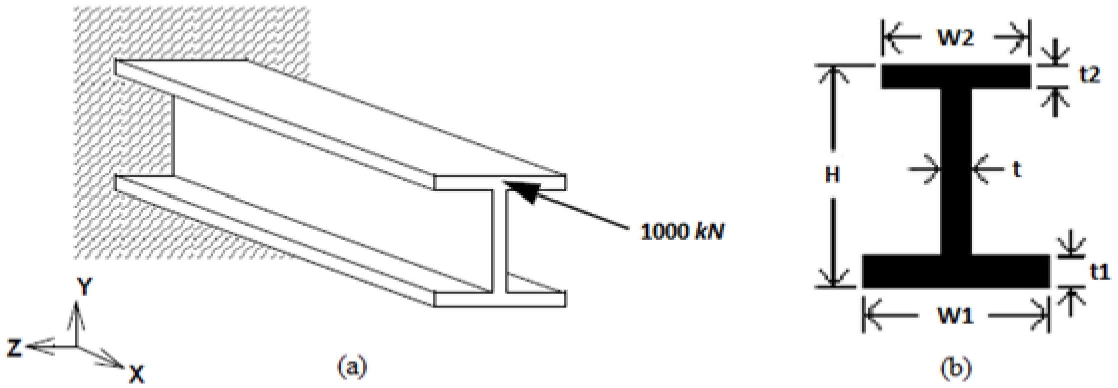


Figure 2. (a) Wide flange beam, W460x74 subjected to an eccentric load; (b) Cross section of wide flange beam, W460x74 [18].

Table 1. Wide flange beam W460x74 dimensions and its properties.

Dimension	
H (mm)	457.00
t (mm)	9.00
t1 (mm)	14.50
t2 (mm)	14.50
W1(mm)	190.00
W2 (mm)	190.00
Properties	
Young's Modulus (N/mm^2)	200000
Poisson ratio	0.3

4. Model Validation

The FE models of the wide flange beam W460x74 are validated by comparing the results of FEA with theoretical calculation. Table 2 shows the comparison between FEA results with theoretical calculation in terms of stresses and displacement. Theoretical values of stresses and displacement are calculated using Eq. (1-6) (Beer et. al., 2006) with dimensions and properties as mentioned in Table 1. The purpose for comparison is also to ensure that the FE model simulated is valid in terms of loading and boundary conditions. It can be seen in Table 2 that the results between the FE models are in good agreement with the theoretical values with acceptable errors. Therefore, all the FE models are considered valid and can be used for this study.

Normal stress:

$$\sigma_n = \frac{F}{A} \tag{1}$$

Bending stress:

$$\sigma_m = \frac{Mc}{I} \tag{2}$$

Maximum combined stress:

$$\sigma_T = \frac{F}{A} + \frac{Mc}{I} \tag{3}$$

Deflection from axial load:

$$\delta_n = \frac{FL}{AE} \tag{4}$$

Deflection from bending moment:

$$\delta_m = \frac{ML^2}{2EI} \tag{5}$$

Maximum deflection:

$$\delta_T = \sqrt{(\delta_n)^2 + (\delta_m)^2} \tag{6}$$

where, F is the axial load,
M is bending moment,
A is the I-beam cross sectional area,
I is the I-beam second moment of area,
σ_n is the normal stress,
σ_m is the bending stress,
σ_T is the maximum stress due to the applied load,
δ_n is the deflection from axial load,
δ_m is the deflection from bending moment,
δ_T is the maximum deflection due to the applied load.

Table 2. Comparison between FEA and theoretical calculation of stresses and displacement for wide flange beam W460x74.

Results	Theory	FE Model		
		1D	2D	3D
Maximum Tensile Stress (MPa)	32.251	32.300	33.950	26.650
Maximum Compressive Stress (MPa)	245.881	246.000	252.500	247.500
Maximum Beam Deflection (mm)	38.132	38.100	38.400	38.700

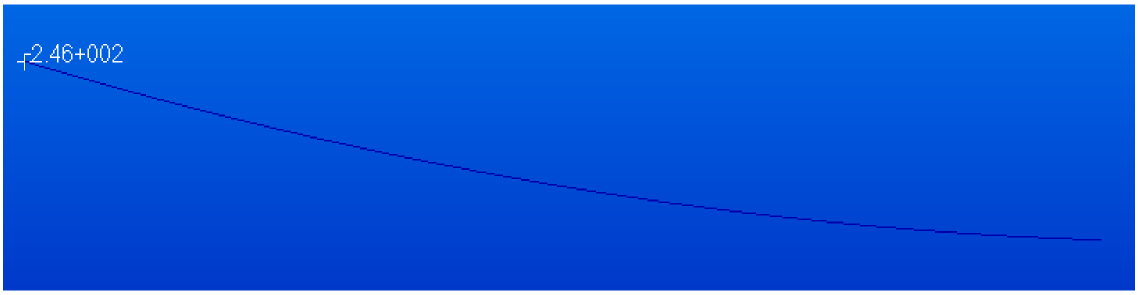
5. Results and Analysis

Table 3 shows the results available for 1D FEA, 2D FEA, and 3D FEA for wide flange beam W460x74. Stress results for 1D FEA are available in the form of bar stresses and are neatly organized into axial stresses, bending stresses, and combined axial and bending stresses. Axial stresses are stresses due to force acting normal to the surface while bending stresses are stresses due to forces which produce bending moment. Combined axial and bending stresses are further divided into minimum and maximum combined stresses. Stresses in 2D FEA and 3D FEA are displayed in the form of stress tensor. Stresses such as axial stresses, bending stresses and combined axial and bending stresses must be perceived and interpreted by the analyst through the stress tensor.

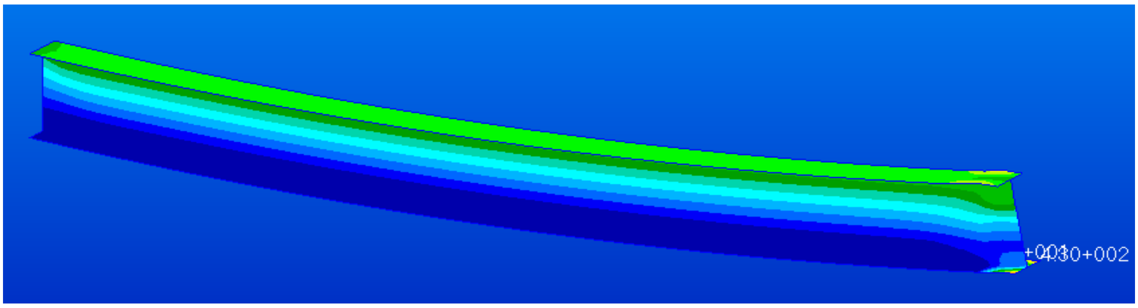
Table 3. Comparison of FEA results between 1D, 2D, and 3D FE model.

Analysis Results	FE Model		
	1D	2D	3D
Stress tensor	NA	Available	Available
Bar stresses, Axial	Available	NA	NA
Bar stresses, Bending	Available	NA	NA
Bar stresses, Max combined	Available	NA	NA
Bar stresses, Min combined	Available	NA	NA
Displacement	Available	Available	Available
Deformation	Available	Available	Available

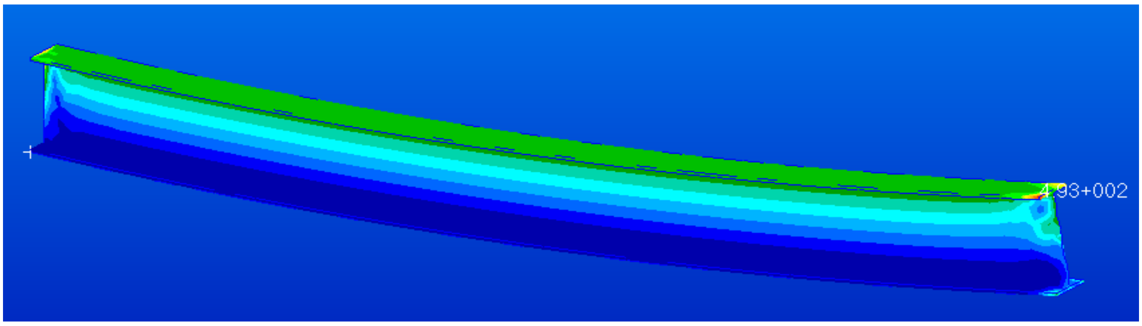
Both displacement results and deformation for 1D FEA, 2D FEA, and 3D FEA are available as resultant or as separate components in the x-axis, y-axis, and z-axis directions. Figure 3 shows the deformation of the I-beam modelled using 1D element, 2D element, and 3D elements respectively for the simple case study. 3D element gives a better illustration of deformation of the I-beam than that of 2D element. 1D element gives the simplest but sufficient illustration of the I-beam deformation as the beam elements are modelled as a line at the neutral axis of the actual beam.



(a)



(b)



(c)

Figure 3. Deformation of wide flange beam W460X74 due to Applied Load: (a) 1D FE model; (b) 2D FE model; 3D FE model.

Table 4 displays the deflection of the beam in the y-axis due to the applied load. The deflection is taken at ten different but constant increment of the length of the beam. Theoretical values are obtained using Eq. (7) [17].

$$\delta_y = \frac{ML}{EI} \quad (7)$$

where, δ_y is the deflection at the end of the beam,
M is bending moment,
L is the length of the beam,
E is the Young's Modulus of the beam,
I is the I-beam second moment of area.

Table 4. Deflection of wide flange beam W460x74 at different beam length.

Length of Beam, x (mm)	Deflection of Beam, y (mm)			
	Theory	1D	2D	3D
500	0.4648	0.4650	0.3520	0.3650
1000	1.6125	1.6100	1.4900	1.5100
1500	3.5159	3.5200	3.3800	3.4100
2000	6.1790	6.1800	6.0400	6.0700
2500	9.6027	9.6000	9.4600	9.4900
3000	13.7869	13.8000	13.6000	13.7000
3500	18.7319	18.7000	18.6000	18.6000
4000	24.4376	24.4000	24.3000	24.3000
4500	30.9040	30.9000	30.7000	30.8000
5000	38.1311	38.1000	38.1000	38.0000

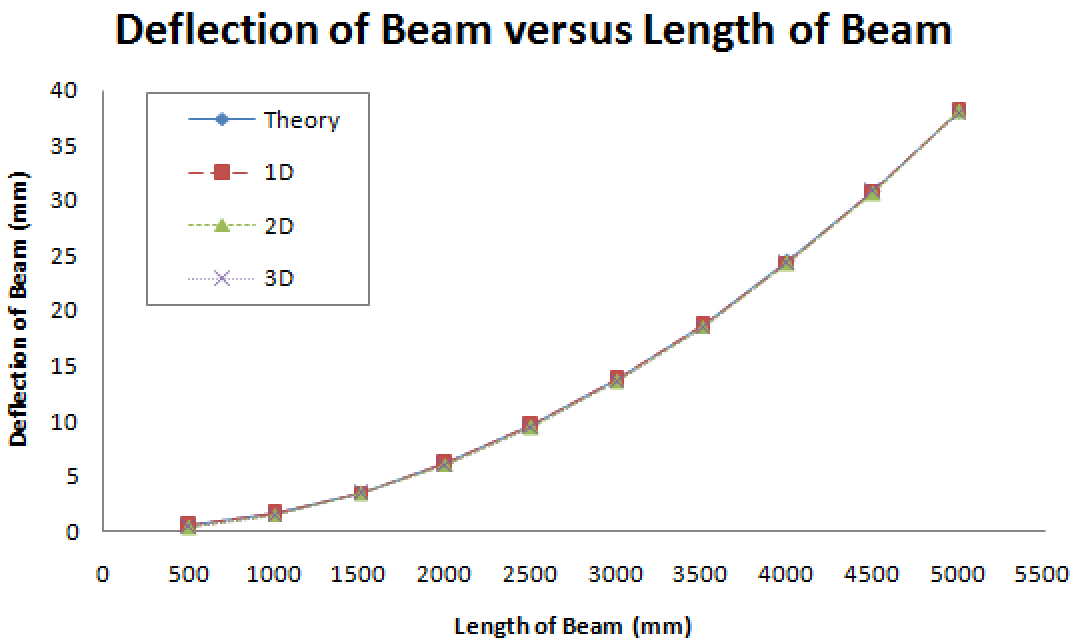


Figure 4. Wide flange beam W460x74 deflection at different length of beam.

6. Discussion

For static analysis in FEA, all types of element will yield the stress and displacement results, which are the two most important parameters in evaluating the structure integrity. However, 1D element provides an easier method of interpretation of FEA results as the results are neatly organized and grouped into individual stresses and displacements as shown in Table 4. 2D and 3D elements, although they provide greater detail, they require some effort in the interpretation of results. The results obtained can be confusing, especially for new users, but if carefully interpreted can be reliable and accurate.

Graphically, 3D element produces the clearest deformation which may be experienced by the beam due to the applied load because the model is simulated closest to the actual I-beam. The importance of the detail results is the existence and location of stress concentration which may cause fatal failure of the structure. This detail stress distribution is only available in 2D FE model and 3D FE model.

In terms of accuracy, 1D element gives the exact stresses and displacements values as those calculated by theory because the assumptions used in 1D element are those of the same as theory. 2D and 3D elements produce stresses and displacement values which deviate slightly from the theoretical values but with acceptable errors. 2D and 3D elements take into account the plane stresses due to shear force between the elements. Due to the consideration of plane stresses in 2D and 3D elements, the results obtained are more detail in terms of stress distribution and displacement.

Since the errors are small and negligible, the choice of elements for FEA depends largely on the geometry of the structure. This is because not all structure can be modelled using 1D element or 2D element. 1D element is used for long and slender structure such as javelins, poles, etc. 2D element is used for plate or shell like structure while 3D element is used for structure with complex geometry which cannot be simplified for analysis. A technique known as dimensional reduction is usually used to reduce the complexity of the structure from 3D or 2D to 2D and 1D respectively. Execution time and computational memory can be saved with this technique.

The type of results desired also plays a role in choosing the right element for FEA. If a detail analysis on the stress distribution to determine the areas of stress concentration is required, then 2D element or 3D element are of better choices. 1D element can still be used for rough estimation and prediction of failure of the structure.

Lastly, the choice of element for FEA also depends on the time and memory capacity of the computer available. If a simple and quick analysis on the structural integrity is required, 1D element is of better choice. If a large computer memory is available, a detail analysis is always the best choice for FEA in which modelling with 2D element or 3D elements is required.

7. Conclusion

The similarity when modelling with 1D element, 2D element, and 3D element is that they are able to generate the same result parameters. For instance, linear static analysis in FEA gives result of stress and displacement though the values might differ from one another. The difference however, does not mean that the results are inaccurate. 1D element will provide a much organize result but less detail which require less effort in the interpretation of results. 2D and 3D FEA results are displayed in the form of tensor, which provide more details for each section of the structure but also require more effort to interpret the results. Besides method of modelling and application of load and boundary conditions, 1D element, 2D element, and 3D element also differ in terms of deformation of the structure. 1D element gives a rough idea of how the structure will deform when load is applied. While 2D element gives a clearer deformation, 3D element gives the best insight to the deformation of structure. With the advantages and limitations in mind, three factors are to be considered when deciding on the types of elements to use in FEA, which are the geometry of the structure, desired results, and time frame as well as the capability of the computer. The most important factor is the geometry of the structure. If the simple, symmetry and uniform such as beam, plate, rod, etc. 1D element is desirable. 3D element should be used only when the structure has complex geometry which cannot be simplified. Next factor to be considered is the results required. 2D element and 3D element are used for detail results such as determination of stress distribution, while 1D element is used for rough estimate. Last but not least is the execution time and memory required for the solver to run the analysis. 1D FEA require the least memory and the fastest to complete while 3D FEA require the most memory and slowest to complete.

8. Recommendation

1D element, 2D element, and 3D element are discussed in general in this study. It would be beneficial to compare and contrast the different types of 1D element, 2D element, and 3D element. The results of the present work are limited to isotropic materials. It is desirable to investigate the effect of material properties on the choice of elements in FEA. Different materials including orthotropic and anisotropic materials such as composites can be used in future study to compare and contrast the result of this study. Three important loadings which may occur in a structure are axial, bending, and torsional loads. The current study only involves axial and bending loads. A case study on the effect of torsional load on the choice of elements would be beneficial to the understanding of the choice of elements used in FEA.

9. References

[1] Wu S R and James C 1997 Advanced development of explicit FEA in automotive applications Computer Methods in Applied Mechanics and Engineering. **149** 189-199

[2] Kamal M, Rahman M M and Rahman A G A 2012 *J Mech Eng Sci* **3** 291-300.

[3] Kamal M, Rahman M M and Rahman A G A 2013 *Inter J Automot Mech Eng* **7** 912-923.

[4] Mahendran M 2007 *Applications of Finite Element Analysis in Structural Engineering* Proc. Int. Conf. on Computer Aided Engineering. 38-46

[5] Zulkifli A, Ariffin A K and Rahman M M 2011 *Inter J Automot Mech Eng* **3** 256-264.

[6] Daud R, Ariffin A K and Abdullah S 2013 *Inter J Automot Mech Eng* **7** 995-1006.

[7] Huzni S, Ilfan M, Sulaiman T, Fonna S, Ridha M and Arifin A K 2013 *Inter J Automot Mech Eng* 7: 1023-1030.

[8] Newberry A L 2000 FEA modeling proves its worth in composite design in RP Asia 2000 Conf. held at Kuala Lumpur Malaysia 20-22 September 2000

[9] Kadirgama K, Rahman M M, Ismail A R and Bakar R A 2011 *J Mech Eng Sci* **1** 37-66.

[10] Najiha M S, Rahman M M, Yusoff A R and Kadirgama K 2012 *Inter J Automot Mech Eng* **6** 766-774.

[11] Department of Aerospace Engineering Sciences University of Colorado 2012 Introduction to finite element methods (ASEN 5007) Ch. 6 FEM Modelling: Introduction. [Accessed 21 December 2012] Available from http://www.colorado.edu/engineering/cas/courses.d/IFEM.d/IFEM.Ch06.d/IFEM.Ch06.pdf

[12] Logan D L 2007 *A First Course in The Finite Element Method 4th ed* (Canada: Thomsom Canada Limited)

[13] Berlioz A and Trompette P 2010 *Solid Mechanics Using The Finite Element Method* (USA: John Wiley & Sons Inc.

[14] Rugarli P 2010 *Structural Analysis with Finite Elements* (UK: Thomas Telford Limited)

[15] Papadopoulos P 2010 ME280A: Introduction to finite element method [Accessed 4 December 2012] Available from www.kochmann.caltech.edu/ae108a/FEnotes.pdf

[16] Boeraeve P 2010 Introduction to the finite element method (FEM) [Accessed 4 December 2012] Available from www.gramme.be/unite9/FEM/Finite%20Element%20Method.pdf.

[17] Beer F P, Johnston Jr E R and DeWolf J T 2006 *Mechanics of Materials 4th ed* (Singapore: McGraw-Hill Education)

[18] MSC Software Corporation MSC Nastran 105 Exercise Workbook: Workshop 13a load analysis of a beam (using a point force and moment).

Acknowledgments

The authors would like to acknowledge CRIM, and Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka (UTeM) for supporting the present work through PJP/2012/FKM(18C)S1112 Research Fund.