The Application of EXCEL in Teaching Finite Element Analysis to Final Year Engineering Students.

Kian Teh and Laurie Morgan

Curtin University of Technology

Abstract.

Many commercial programs exist for finite element analysis, however these are usually quite specialised and hence complicated to use by the uninitiated. On the other hand, some of the programs are extremely user-friendly, to the extent that it is possible for students to obtain the solutions without fully understanding the related theories.

The teaching strategy used is aimed at the group of students who either have not acquired sufficient foundation in programming, or seem reluctant to learn it in this context. Briefly, the strategy involves the matrix manipulation capability of a spreadsheet program such as Microsoft EXCEL.

EXCEL software is chosen as a computer package that offers a middle ground between programming and using packaged FEA software. EXCEL provides an ideal platform to provide a visualisation of the numerical calculations and data processing, to explain the algorithm of assembly of a global stiffness matrix, the implementation of boundary conditions, the solution process, and the influence of mesh pattern on solution convergence. The students are taught the basic principles and conceptual issues of finite element analysis while requiring minimum programming skills. The students must acquire a complete insight of the mathematical context, to be able to develop their own spreadsheet successfully.

Introduction.

The objective of this paper is to report on a teaching strategy that has been adopted for teaching final (4th) year undergraduates in a Mechanical Engineering degree program.

Finite Element Analysis (FEA) has long been part of this program, but the change in the scholarship and expectation of the students, and the rise in quality and ease of use of commercial FEA software, have meant that in order for the education process to continue to provide valuable instruction, the method of teaching has had to change.

Many commercial programs for finite element analysis are readily available, however these are usually quite specialised and hence complicated to use by the uninitiated. On the other hand some of these programs are extremely user-friendly, to the extent that it is possible for students to obtain the solutions without fully understanding any of the related theories.

The teaching strategy used is aimed at the group of students who either have not acquired sufficient foundation in programming, or seem reluctant to learn it in this context.

Briefly, the strategy involves the matrix manipulation capability of the Microsoft EXCEL (or similar) spreadsheet. The Microsoft EXCEL spreadsheet program is readily available to almost all students, and it is a computer package that offers a middle ground between programming and using specialised FEA software. EXCEL provides an ideal platform to provide a visualisation of the numerical calculations and data processing, the algorithm of assembly of a global stiffness matrix, the implementation of boundary condition, the solution process, and the influence of mesh pattern on solution convergence. The students are taught these basic principles and conceptual issues of finite element analysis while requiring minimum programming skills. The students must however, acquire a full insight of the mathematical context to be able to develop their own spreadsheet successfully.

The spreadsheet approach allows the students to focus on thinking and comprehension of the subject matter at hand rather than on the need for programming. However, this approach limits the students to solving simple problems. A general-purpose commercial finite element method software is introduced later to further demonstrate the applications of finite eleme nt analysis to more complicated problems.

This teaching strategy was first introduced in 2004 and received encouraging response from the students, especially those who were weak in writing computer programs. It was decided to continue this teaching strategy for the second time this year.

Background.

The objective of this section is to outline the mathematical relationships involved in the static analysis of a plane structures using the finite element method[1],[2].

The Finite Element Method (FEM) is a numerical method that involves basic matrix algebra, as the governing equations of equilibrium are normally established in a matrix form. Often simple structural problems are used to introduce this numerical method, and their solution usually requires difficult numerical manipulation, which is a laborious, and error prone task. EXCEL provides a very good platform to take over the ‘number crunching’ process and the students are free to focus on the understanding of the basic algorithm of the numerical method.

A brief review of the finite element method.

This section presents the essential mathematical relationships that are required for the illustrative example described in a later section.

The fundamental assumption adopted in the finite element method is that the structure can be represented by the assembly of a finite number of elements, these elements being interconnected so as to represent the actual structure. The boundary conditions are compatible at the nodal points where the elements are joined. The element forces $\{F\}_e$ are related to the corresponding displacements $\{u\}_e$ by the matrix equation

$$\{F\}_e = [k]_e\{u\}_e, \qquad (1.0)$$

where $[k]_e$ is generally referred to as the element stiffness matrix. For each element in the structure, the element stiffness matrix is calculated using equation (1.0).

In order to determine the stiffness matrix of the complete structure, a common datum must be established for all unassembled element stiffness matrices so that all the forces and their corresponding displacements will be related using a common coordinate system usually called the global coordinate system.

Equation (1.0) is written in terms of a local coordinate system, and this equation can be rewritten in terms of the overall, global datum coordinate system, as:

$$\{F\}_g = [k]_g\{u\}_g, \qquad (2.0)$$

where: $\{F\}_g = [T]\{F\}_e$, $[k]_g = [T]^T[k]_e[T]$ and $\{u\}_g = [T]\{u\}_e$. [T] is the transformation matrix relating the local and global coordinate systems.

The stiffness matrix for the complete structure as a free body is obtained by assembling all the global element stiffness matrices, computed using equation (2.0), to form a single matrix. In order to determine the nodal displacement due to an externally applied force, this matrix must be modified to reflect the homogeneous boundary conditions. This is accomplished by deleting corresponding rows and columns from the complete stiffness matrix. Only then is it possible to obtain the solution for displacements from $\{u\}_r = [K]_r^{-1}\{F\}_r$, where the subscript r is used to indicate that all matrices have been reduced in size to exclude forces and displacements at the selected points.

Comments on the capability of some of the built in commands from EXCEL

The EXCEL spreadsheet is a very popular accounting software that is designed to do statistical calculations, database operations and sort lists of information. It has sufficient engineering functions and formulae to be very useful for engineers and scientists to perform engineering calculations[3].

In the case of implementing the finite element method, EXCEL has the following built-in functions that are very useful.

1. Trigonometric functions:
   - **radians** (angle): converting angles in degrees to radians,
   - **cos**(number) and **sin**(number): computing the cosine and sine of an angle.

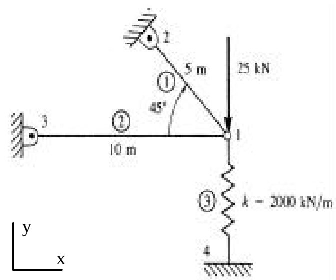
2. Array formulae : although the matrix algebra operations available in EXCEL are limited, there are enough array formulae to perform the necessary matrix manipulations. The array formulae used are;
   - **MINVERSE**(array): This formula calculates the inverse of a matrix stored in array and displays the inverse in the highlighted range of cells.
   - **MMULT**(array): This returns the matrix product of two arrays,
   - **TRANSPOSE**(array): This returns the transpose of a matrix,

It is important to note that when using the array formulae one has to know the size of the matrix that the result will produce. EXCEL handles array formulae differently to normal formulae. An array formula must be entered in a special way: select the cell or cells that will contain the formula, create the formula, and then press CTRL+SHIFT+ENTER to enter the formula.

Illustrative example

Referring to an example that is used in the authors' teaching will best illustrate the application of EXCEL to assist the teaching of the finite element method.

Figure 1 shows a plane truss structure with a spring and a concentrated load acting on node1.



Global coordinate system

The structure can be represented by two tensile elements and a spring element. The orientation of the global x-y coordinate system is as indicated, and the global element stiffness for each element can be calculated from the expression $[T]^T[k]_e[T]$, where [T] is the transformation matrix and $[k]_e$ is the element stiffness matrix established with respect to the local coordinate system.

The transformation matrix $[T] = \begin{bmatrix} \cos\theta & \sin\theta & 0 & 0 \\ -\sin\theta & \cos\theta & 0 & 0 \\ 0 & 0 & \cos\theta & \sin\theta \\ 0 & 0 & -\sin\theta & \cos\theta \end{bmatrix}$.

This matrix is implemented in EXCEL with the built in trigonometric formulae, **cos**(number) and **sin**(number), where "number" is the input value for angle in radians.

The element stiffness matrices for the tensile element and the spring element both have the same format: $[k]_e = \begin{bmatrix} k & 0 & -k & 0 \\ 0 & 0 & 0 & 0 \\ -k & 0 & k & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$

For a spring element k is the spring stiffness, for a tensile element the stiffness k is calculated from the cross section area, elastic modulus and length in $\frac{AE}{L}$.

Since the expression $[T]^T[k]_e[T]$ is constantly referred to each time there is a need to calculate the global element stiffness matrix, a separate EXCEL worksheet is setup for the purpose of performing the necessary matrix algebra (refer to Figure 2). The required input data is passed into this work sheet and is linked to the active worksheet.

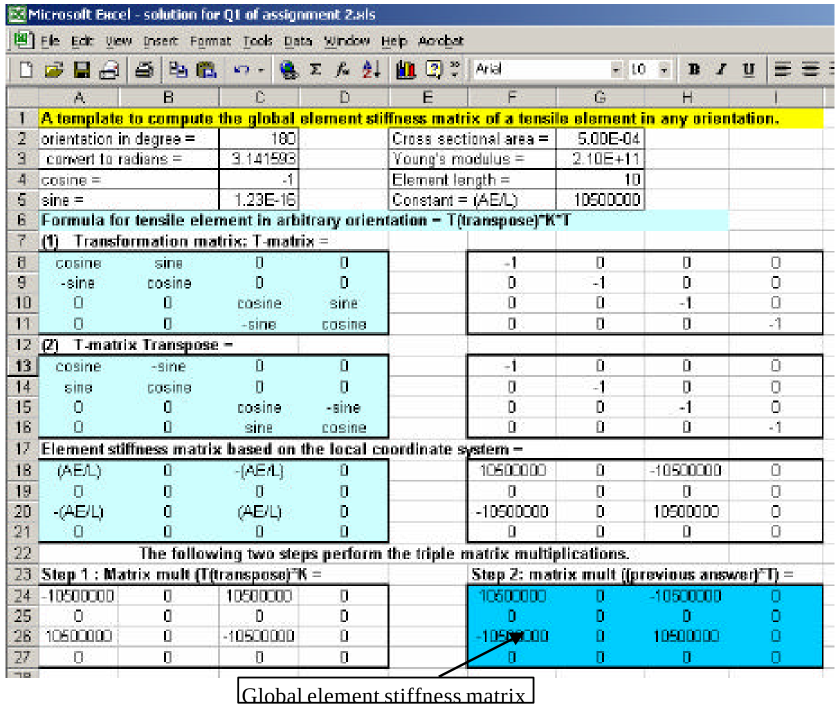


Figure 2 shows the template for calculating the global element stiffness matrix $[k]_g$ of a tensile element according to the equation $[k]_g = [T]^T[k]_e[T]$, where $[k]_e$ is the local element stiffness matrix and [T] is the transformation matrix.

The following results are obtained from the template as shown in Figure 2.

The global element stiffness matrix for first tensile element is calculated based on the following data

- Element orientation taken anticlockwise with respect to the horizontal axis = 135^0
- Cross sectional area = 0.0005 m^2
- Element length = 5 m
- Young's modulus = 210 GPa.

10500000	-10500000	-10500000	10500000
-10500000	10500000	10500000	-10500000
-10500000	10500000	10500000	-10500000
10500000	-10500000	-10500000	10500000

(3.0)

The global element stiffness matrix for second tensile element is calculated based on the following data,

- Element orientation taken anticlockwise with respect to the horizontal axis = 180^0
- Cross sectional area = 0.0005 m^2
- Element length = 10 m
- Young's modulus = 210 GPa

10500000	0	-10500000	0
0	0	0	0
-10500000	0	10500000	0
0	0	0	0

(4.0)

The global element stiffness matrix for the spring element is calculated based on the following data,

- Element orientation taken anticlockwise with respect to the horizontal axis = 90^0
- Spring stiffness = $2x10^6$ N/m.

0	0	0	0
0	2000000	0	-2000000
0	0	0	0
0	-2000000	0	2000000

(5.0)

Assembly of global stiffness matrix.

One of the difficult concepts experienced by the students is the assembly of the global stiffness matrix from the global element stiffness matrices. This feature can be best explained by using different colours to illustrate the corresponding locations of the various terms in the element and global stiffness matrices.

For example the global element stiffness of the second tensile element is shown below, and the associated nodal degrees of freedom, u1, v1, u2 and v2, are also included.

10500000	0	-10500000	0	**u1**
0	0	0	0	**v1**
-10500000	0	10500000	0	**u2**
0	0	0	0	**v2**

(6.0)

The terms in the matrix have been partitioned and coloured to indicate their association with the respective nodal degree of freedom. The matrix can be expanded to indicate the corresponding locations in the global stiffness matrix.

10500000	0			-10500000	0			**u1**
0	0			0	0			**v1**
								u2
								v2
-10500000	0			10500000	0			**u3**
0	0			0	0			**v3**
								u4
								v4

(7.0)

where u1,v1,…,u4,v4 are the global nodal degrees of freedom.

This treatment is repeated for the remaining elements, and the global stiffness matrix is obtained by summing all three expanded global element stiffness matrices.

21000000	-10500000	-10500000	10500000	-10500000	0	0	0	u1	
-10500000	12500000	10500000	-10500000	0	0	0	-2000000	v1	
-10500000	10500000	10500000	-10500000	0	0	0	0	u2	
10500000	-10500000	-10500000	10500000	0	0	0	0	v2	
-10500000	0	0	0	10500000	0	0	0	u3	(8.0)
0	0	0	0	0	0	0	0	v3	
0	0	0	0	0	0	0	0	u4	
0	-2000000	0	0	0	0	0	2000000	v4	

The geometric boundary conditions of the structure shown in Figure 1 dictate that all the degrees of freedom at nodes 2, 3 and 4 have zero value, i.e. u2 = v2 = u3 = v3 = u4 = v4 = 0. Implementation of these boundary conditions is achieved by simply deleting rows 3 to 8 and their corresponding columns, thus reducing the 8x8 global stiffness matrix to a 2x2 matrix.

To implement these conditions in EXCEL is extremely easy. It is best that the whole global stiffness matrix is copied onto a new worksheet, then the following operations can be performed without interfering with the active worksheet. To delete a column in a worksheet simply click on the respective column letter and then click on the delete command in the Edit menu, refer to Figure 3. This operation causes the matrix to automatically reduce in size.

To delete a row, these operations are repeated by clicking on the appropriate row number and then selecting the delete command in the Edit menu. The reduced global stiffness matrix can then be copied and pasted back to the active wo rksheet to continue with the calculation.

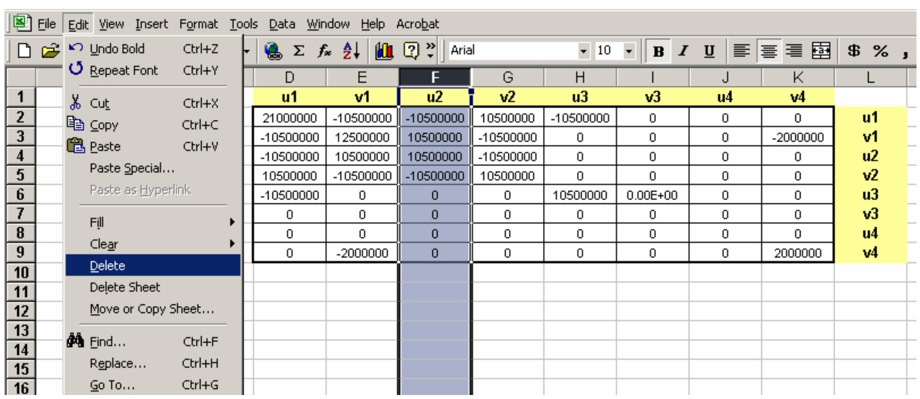


Figure 3 shows the operations for the automatic reduction of the number of columns of the matrix.

Figure 4. Following the process shown in the previous figure, the 8x8 global stiffness matrix is reduced to a 2x2 matrix after deleting appropriate rows and columns corresponding to the homogeneous boundary conditions.

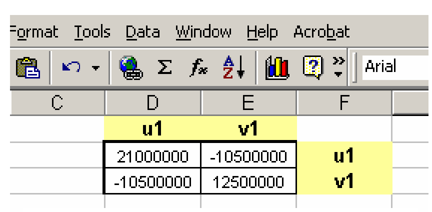


Figure 4. Reduced global stiffness matrix.

After implementing the boundary conditions, the resulting force-displacement relationship can be written as,

$$\begin{Bmatrix} F_{x1} \\ F_{y1} \end{Bmatrix} = \begin{bmatrix} 21\times10^6 & -10.5\times10^6 \\ -10.5\times10^6 & 12.5\times10^6 \end{bmatrix} \begin{Bmatrix} u1 \\ v1 \end{Bmatrix} \qquad (9.0)$$

$$\text{or } \{F\}_r = [k]_r\{u\}_r \qquad (10.0)$$

The unknown displacements at node 1, $\{u\}_r$, can be calculated by inverting $[k]_r$ and then determining the matrix product of $[k]_r^{-1}$ and $\{F\}_r$.

Inversion of the reduced global stiffness matrix $[k]_r$ using EXCEL is shown in Figure 5. Since the inverted matrix is of size 2x2, four cells must be highlighted before creating the array formula, and then press CTRL+SHIFT+ENTER to enter the formula.

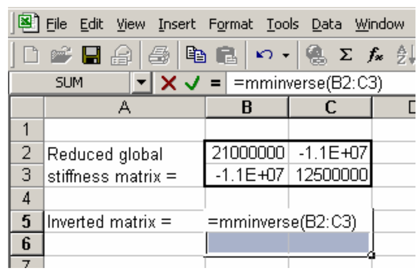


Figure 5: Matrix inversion.

Having determined the inverted reduced global stiffness matrix, The unknown displacements at node 1, $\{u\}_r$, can be calculated by determining the matrix product of $[k]_r^{-1}$ and $\{F\}_r$. Figure 6 shows the operations required for calculating the matrix product of two matrices. The expected outcome is a 2x1 matrix, thus two cells are highlighted, the array formula is created, and pressing CTRL+SHIFT+ENTER completes the operations.

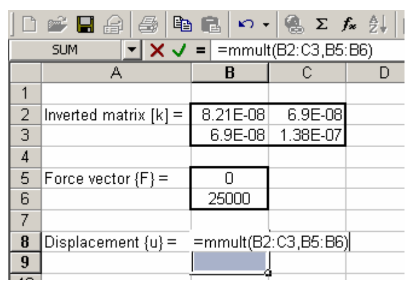


Figure 6: Matrix multiplication.

Discussion

The illustrative example shows the application of the finite element method, using an EXCEL spreadsheet approach, to calculate the deformation of a plane truss structure subjected to a concentrated load, (see Figure 1). The solution algorithm of this numerical method is easily performed and explained to the students using the spreadsheet software, in particular the matrix algebra. The spreadsheet approach enables the students to spend their time working directly with the numerical problem rather than on constructing a computer program. In other words, the use of spreadsheet programs can provide the student with additional insight into the principles involved in the finite element method, without getting bogged down with the programming aspects of the problem solution.

Most students are familiar with EXCEL and have used this software for data management and result presentation for laboratory reports in other subjects. For some students the matrix algebra capability is new to them, however, it does not take them long to grasp and use these array formulae.

Having established the solution spreadsheet, it is then possible to use this spreadsheet to explore other issues such as the effect of different load systems acting on the plane truss structure. This is done simply by replacing the reduced force vector, equation (10.0), with a new set of values, and EXCEL recalculates the nodal displacements automatically.

Another advantage of using the spreadsheet approach is the ease of studying the effect of modeling a structure using different size elements, i.e. the effect of different mesh density. Another example using beam elements is used to introduce this concept. The need for uneven mesh density arises when there is a stress concentration present in the structure. Previously it was difficult to deal with this topic in the class due to the large number of numerical calculations. However, with this spreadsheet approach it is relatively easy to illustrate to the students the calculation of the element stiffness matrices, the assembly of the global stiffness matrix, the implementation of the boundary conditions, and the solution for the unknown nodal displacements.

Conclusion

This spreadsheet approach to introduce the solution algorithm of the finite element method is well received by students. The general comments are

- This approach takes away the large amount of number crunching task, that is usually tedious and error prone.
- A "program" can be set up using the spreadsheet software, which helps the students to gain a better understanding of this numerical method.
- When using a commercial finite element program, one does not get the same learning experience regarding the solution algorithm. Some commercial FEA software makes it all seem too easy.
- The colour feature in EXCEL makes it easy to present and highlight certain important matters.
- The spreadsheet has introduced visualisation to the solution algorithm which is otherwise difficult to attain in the traditional pen and paper approach.
- More capable students have taken the initiative to introduce macro-programming to replace some of the repetitive cut-and-paste operations.

References

[1] R.D. Cook, D.S. Malkus and M.E. Plesha, *Concepts and Applications of Finite Element Analysis*, John Wiley & Sons, 3rd edition, 1974.

[2] D.L. Logan, *A First Course in the Finite Element Method,* Brooks/Cole, Thomson Learning, 3rd edition, 2002.

[3] S.C. Bloch, *EXCEL for Engineers and Scientists*, John Wiley & Sons, 2003.

KIAN TEH is a senior lecturer in the Department of Mechanical Engineering at the Curtin University of Technology. He received his Ph.D. in Mechanical Engineering from Melbourne University. His research interests are in applications of finite element method to design and the effective use of technology in education.

LAURIE MORGAN is a senior lecturer in the Department of Mechanical Engineering at the Curtin University of Technology. Mr Morgan has practiced many years as a mechanical engineer and he lectures in mechanical design units in the undergraduate programs. His research interests are application of new technology in mechanical design and engineering education.