Ramsay Maunder ASSOCIATES *Finite Element Specialists and Engineering Consultants*

The fib Model Code for Concrete Structures 2010
A Critique of the Comments on Finite Element Analysis

Introduction

The following extracts from the *fib* website explain the history of the organisation and the rational of their 2010 publication:

> The International Federation for Structural Concrete (*fib* - Fédération internationale du béton) is a not-for-profit organisation created in 1998 from the merger of the Euro-International Committee for Concrete (CEB - Comité euro-international du béton) and the International Federation for Prestressing (FIP - Fédération internationale de la précontrainte). The parent organisations CEB and FIP existed independently since 1952…
>
> In October 2013, the hardcover and e-book editions of the *fib Model Code for Concrete Structures 2010* (MC2010) were published by the Ernst & Sohn publishing house. This milestone publication presents new developments and ideas with regard to concrete structures and structural materials, and will serve as a basis for future codes for concrete structures. It is as an essential document for national and international code committees, practitioners and researchers.
>
> http://www.fib-international.org/fib-model-code-2010

In section 7.11.2.2 of this document a description of the Finite Element Method is provided together with some basic guidance and words of caution. Having cast an eye over the description/guidance we thought that a few additional words might help the discerning reader to better understand some of the statements made in the document.

Fib Description of FE Method

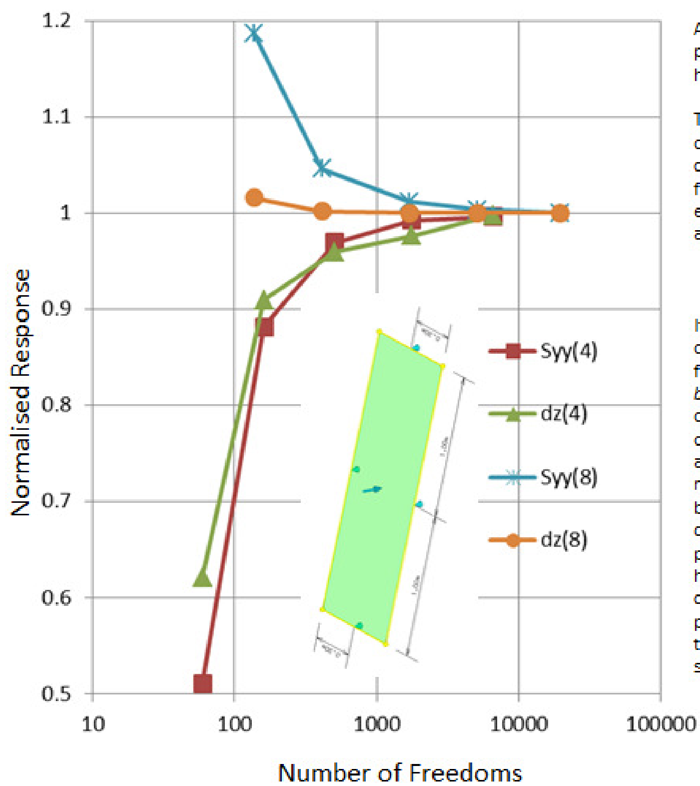
"The finite element method is typically used for the numerical solution of continuum problem. Depending on the type of formulation (stiffness, compliance and mixed methods) the results are by definition different from the exact solution. In case of the most widely used stiffness method, the shape of displacement field is assumed and equilibrium is satisfied only in integral sense. That means, the best possible equilibrium is found for a given approximation (finite element type and size). ***The internal stresses are lower***, compared with an exact solution. A finite element formulation should satisfy the requirement of convergence to the exact solution by reducing the element size (and increasing the number of degrees of freedom). Thus, independently of the material model, the approximations introduced by the finite element formulation only, can be a significant source of errors in numerical analysis. Similarly, in other numerical methods, the errors due to approximations should be adequately checked."

RMA Description of FE Method

The finite element method is probably the most popular method adopted for the numerical approximation of continuum problems. For well-posed engineering problems, the method is capable, with sufficient mesh refinement, of capturing the theoretically exact solution. If the mesh is not sufficiently refined then the solution is approximate and the nature of the approximation is dependent on the finite element formulation adopted. In particular, for the most commonly used pure displacement element formulation, the approximation manifests itself in finite element stresses that are only weakly in equilibrium with the applied loads. This poor approximation of the true stress is potentially a significant issue for the engineer since without equilibrating stresses he/she is unable accurately to judge the required material capacity for a structure – for a classic example with catastrophic consequences see http://en.wikipedia.org/wiki/Sleipner_A.

It is incumbent therefore on the practicing engineer and his/her manager that some form of simulation governance, or verification, is conducted so that the results produced are deemed of sufficient accuracy so as to be able to accurately judge the required material capacity for the structure.

For pure displacement element models loaded with applied forces, the linear elastic finite element strain energy will be lower than the true value but should converge towards the true value as the mesh is refined. This implies that the approximate finite element model is over-stiff and that both deflections and stresses will, on average, be less than the true values. Conventional finite element systems however often employ elements that have been formulated in a variety of different ways to overcome some of the shortcomings of pure displacement elements. The results from such elements may not behave in the same manner and can indeed show performance that is over-flexible. A good example of such behaviour is that occurring for the lower-order and higher-order plate elements in ANSYS. Whereas the lower-order element shows results typical of a pure displacement element, i.e. strain energy, displacement and stress all converging from below the true values, the higher-order element shows the opposite behaviour – see figure 1. It is therefore unwise, without further information or experimentation, to assume that the results from a conventional displacement finite element system will converge as predicted.



A 2m by 0.6m steel plate of thickness 0.01m is considered. The plate is simply supported with a UDL of 100kPa. The material has a Poisson's Ratio of 0.3.

The maximum lateral displacement (dz) and direct stress (Syy) occur at the centre of the plate and this figure shows the convergence of these quantities with mesh refinement for both four-noded (SHELL181) and eight-noded (SHELL281) shell elements. The values have been normalised by the values achieved with the most refined eight-noded element mesh:

Syy = 262MPa and dz = 8.459mm

It is interesting to note that the direction in which convergence occurs is dependent on the element type with results converging from *above* the true solution for the eight-noded element and *below* the true solution for the four-noded element. There is an often presented argument that as displacement elements are over-stiff then under applied forces they will tend to deform less and therefore induce less stress than would occur in the actual member. This is of course true in an integral sense and this can be seen in the strain energy convergence for a pure displacement element model under applied forces. In terms of point as opposed to integral values of quantities it is possible however for individual points to behave differently. It is also the case that some elements used in commercial FE systems are not pure displacement elements and so for a number of reasons these arguments are spurious and may not generally hold as seen with this example.

Figure 1: Finite element results for a simply supported rectangular plate under a UDL