1. INTRODUCTION

The finite element method of analysis has undergone considerable development over the past thirty years. This has led to the production of a large number of computer programs, many of which are now commercially available. Consequently the technique is now widely used in industry, both in the design process and for solving vibration problems of existing hardware.

However, it has been known for different programs to give different solutions to the same problem. This is illustrated in Figure 1 which shows the variation of the frequency of the first bending mode of a twisted cantilever plate as a function of the angle of twist. Analyses were carried out by sixteen different establishments using both finite element and analytical methods. Figure 1 represents a subset of those results presented in reference [1]. Plots 1 and 2 were obtained using a triangular facet shell element, the first with a consistent mass matrix and the second with a lumped mass matrix. Plots 3 and 4 were both computed with quadrilateral elements, the first being a facet shell and the second a doubly curved shell. Both used a lumped mass representation. Plot 5 was obtained using a super-parameteric thick shell element with a consistent mass matrix. Plots 6 and 7 were computed using 8 and 16 node superparametric solid elements having lumped and consistent mass matrices respectively. These represent the least successful of the analyses. Several gave frequencies close to the experimental measurements. The percentage difference between the average of all finite element analyses and the experimental measurements are 3.22, 3.24, 4.56, 3.21 and 9.30 for the five angles of twist considered.

It is also possible for different analysts to use the same program for the same problem and produce different results. Possible causes of this are different choices of idealisation and/or element types and incorrect data. Such problems can be overcome by users' obtaining a better understanding of both the finite element method and the program they use.

There is, therefore, a need for a procedure which will enable a user to test the accuracy of the chosen program which will, at the same time, educate him/her in its capabilities. This paper proposes a procedure for checking the dynamic capabilities of existing programs. The main emphasis is on predicting the frequencies and modes of free vibration of a structure. Also, three different formulations of fluid-structure interaction analysis are compared.
2. PROPOSED PROCEDURE FOR FREE VIBRATION ANALYSIS

The finite element displacement method involves assuming a variation of displacement within each element in terms of the unknown nodal displacements. If these displacement functions satisfy the following conditions, then the solution will converge as the number of elements is increased.

(i) Be linearly independent
(ii) Be continuous and have continuous derivatives up to order \((p-1)\) both within the element and across element boundaries. \((p\) is the order of the highest derivative in the energy expressions). An element which satisfies this condition is referred to as a "conforming" element.
(iii) If polynomial functions are used, then they must be complete polynomials of at least degree \(p\). If any terms of degree greater than \(p\) are used, they need not be complete. However, the rate of convergence is governed by the order of completeness of the polynomial.
(iv) Satisfy the geometric boundary conditions.

Many elements in commercially available finite element programs do not satisfy all of these conditions. In an attempt to increase accuracy and reduce the amount of computing, many modifications are introduced which result in non-conforming elements. Examples of such procedures include the use of reduced integration, the addition of extra internal shape functions, the introduction of numerical parameters which are adjusted to give accurate solutions, and the use of assumed stress distributions, to name but a few. The theoretical manuals provided by program developers tend not to give full details of the element formulation. This necessitates a procedure for checking their accuracy which does not require such details.

It is proposed that the following procedure should be used for validating finite elements models for linear dynamic analysis.

1. Apply a set of validation tests for linear static analysis in order to validate the element stiffness matrix.
2. Apply a set of validation tests to the element inertia matrix
3. Apply suitable benchmark tests.

2.1. Static validation tests

The idea of a set of static validation tests is not a new one. A number of papers have been published on this topic many of which are listed in Reference [2] which presents the current state of the art for membrane and plate bending element. Reference [3] discusses the problems involved in setting up a similar procedure for shell elements.

Reference [2] proposes the following set of tests:

(i) Single element completeness tests
(ii) Completeness tests for a patch of elements
(iii) False zero energy mode tests
(iv) Invariance tests
(v) Single element, shape sensitivity tests
(vi) Benchmark tests

These tests have been applied to the elements of a number of commercial finite element systems in References [4, 5, 6]. Reference [7] proposes benchmark tests for membrane, plate bending, shell and folded plate problems.