Purdue University
Purdue e-Pubs

International Compressor Engineering Conference
School of Mechanical Engineering

1978

The Use of Finite Element Method for Stress Analysis of Compressor Valves

D. Lal

THE USE OF FINITE ELEMENT METHOD FOR STRESS ANALYSIS OF COMPRESSOR VALVES

D.N. Lal, Mechanical Engineer
Carrier Corporation, Research Division
Carrier Parkway, Syracuse, NY 13221

Simple static stress and dynamic analyses of a cantilever type suction valve using easily accessible finite element codes SUPERB and ANSYS are described. This type of study is shown to be of unique value in the preliminary design stages of compressor valves.

INTRODUCTION

The valve, suction or discharge, is one of the most critical components of a compressor. A flapper valve is required to have high flexibility to allow unrestrained fluid passage through the ports for achieving high efficiency and capacity of the compressor, but at the same time it is also expected to have enough stiffness to return back in time to seal the ports completely. The motion subjects the valve to severe cyclic stresses and strains. To make the situation worse, most of the valves have irregular geometry as unavoidable design requirements. This increases the possibility of localized stress concentration and premature failure by fatigue. Consequently, it is imperative for the designer to be able to calculate or measure static displacements, stresses and strains at all critical points of the valve. Further, he also needs to know the dynamic characteristics (natural modes and frequencies) in order to avoid resonance conditions and achieve the desired motions.

An analytical solution of the valve stress problem is difficult and cannot promise good accuracy because of the complex geometry and loading situations. The recently developed Finite Element Method (FEM) of structural analysis (1,2) appears to be a suitable technique for achieving the above mentioned goals. The method provides approximate numerical solutions of adequate accuracy in many cases for the above parameters. The most obvious advantage is that it gives stress solutions for all desired locations on the valve, and a prior knowledge of critical areas is not essential. Furthermore, the analysis can be carried out and material and geometrical variables studied immediately during the design stage.

Some specialized programs have been written for determining dynamic stresses in compressor valves (3), but these are complicated, may require computer simulation of the compressor, and are inaccessible to an average design engineer. The use of general purpose proprietary programs is therefore desirable. In this paper, the use of SUPERB and ANSYS programs available from Structural Dynamics Research Corporation, 5729 Dragon Way, Cincinnati, Ohio 45227, has been demonstrated for a typical cantilever type leaf suction valve. This program is very suitable to valve problems because of the availability of isoparametric shell elements.

THE SUPERB PROGRAM

The choice of SUPERB over other equally popular programs like ANSYS, STARDYNE, NASTRAN, etc., is mainly based on the availability of multiply curved isoparametric thin shell elements (1,2). These elements are superior to conventional elements in two significant ways. Firstly, their assumed displacement function is of a higher order (primarily second and third order) so that linearly or quadratically varying strain fields can be described more accurately using fewer elements. Secondly, the real element geometry is conformally mapped into a simpler geometry for computational convenience, using coordinate interpolation functions (also called "shape functions") which can also be of a higher order; e.g., parabolic (second order) and cubic (third order). This feature allows the use of an element which has curved sides and also curved surfaces, thus approximating the true geometry more closely with fewer elements. The program allows both static and dynamic analyses to be performed on the same geometric model with the substitution of a few input cards. However, since the SUPERB dynamic capability is a new release at this writing, it may have several hidden sources of inaccuracy. For comparison purposes, therefore, the ANSYS program was also used for dynamic analysis.

STATIC ANALYSIS

In reality, the valve motion is a dynamic problem where different modes, if excited, can interact with each other and present a complex stress pattern. Nevertheless, for guidance in the preliminary design stages as to the maximum displacements and stresses produced by the maximum pressure differential applied on the valve, a static finite element analysis can be extremely convenient and useful. Once selected and generated, the same geometrical model can also be used later for the dynamic analysis.

Model Geometry

Figure 1 shows the outline of a cantilever type suction valve resting on the valve plate. It is clamped along the arc LON and has a support plate S underneath. The circles C, D, and E denote the overlying fluid port outlines.

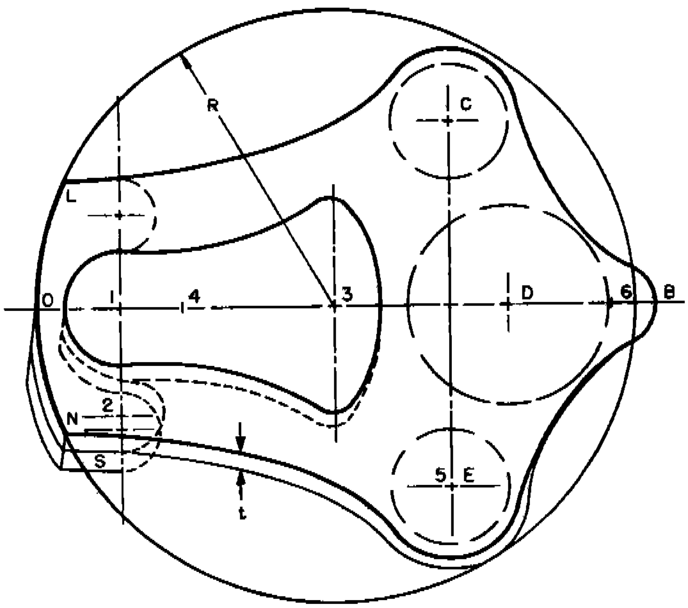

Figure 1. Suction valve below the valve plate

Figure 2 shows the finite element model of the valve using SUPERB. Taking advantage of one fold symmetry, only half of the valve below the horizontal line OB needs to be modeled. The finite element mesh was created using isoparametric parabolic thin shell elements (NSTIF = 7)* (4). This shell element allows both bending and membrane stresses to exist and is defined by nodes on the mid-surface and corresponding thicknesses. The parabolic or second order elements (one intermediate node on the sides) are probably the ideal elements to use because they offer all the advantages of isoparametric formulation at a moderate cost. The third order or cubic elements may offer a slightly better accuracy over the former, but the cost becomes too high to justify its usage in most applications. It should be further noted from Figure 2 that the shape of the elements of the valve near the clamped region was selected to follow the contours of the support plate S, underneath. This configuration enables us to take advantage of some of the built-in program features like automatic mesh generation and convenient coupling of the displacements of overlapping nodes of the valve and the support.

*NSTIF is a computer code name in the SUPERB finite element library to identify the available element formulations.

Six local coordinates at positions 1 to 6 in Figure 1, were defined for convenient and accurate description of most of the curved boundaries. It is seen that only 14 elements cover the entire structure. It is also worth mentioning that a relatively larger number of elements has been used near the deep cutout area M, a possible site for stress concentration. Nodes 51 and 65 represent the centers of ports D and E, respectively. The distributed load or fluid pressure was assumed concentrated on these nodes directed vertically downwards. As is usually a common practice in valve design, the crankcase was provided with a stop to limit the displacement of the valve tip, Node 53, in this case to a maximum of 0.109 inches (2.77 mm).

Loads

Since the actual pressure loads on the valve are unknown, several load steps with increasing magnitude were applied until the valve tip just touched the stop. The valve deformation pattern should change after this threshold load condition, and a few load cases above this threshold were used to reveal any changes in the stress distribution. For input in the program, the fluid pressure in psi was converted to a point load by multiplying with the respective area of the port.

Boundary Conditions

The clamped edge was fixed in all the six allowed degrees of freedom for a node; i.e., the three translations and the three rotations. The cut surface along the central line OB was given appropriate symmetry conditions. The valve tip, Node 53, was left free for loads less than the threshold and constrained to 0.109 inches (2.77 mm) for loads larger than the threshold.

The Static Solution

Figure 3a shows the computer generated geometry plot of the model. This feature is useful in checking the accuracy of the geometrical part (nodal coordinates and elemental connectivities) of the data input.

The finite element solutions giving the nodal displacements and maximum principal stresses are probably the most directly usable quantities for the designer. These can be computer plotted for convenience or hand plotted if desired. Figure 4 shows U_z displacement hand plots of the nodes along the upper edge of the valve model. The curves are shown for three pressures: 5 psi (34.5 KPa), 10 psi (69 KPa), and 60 psi (414 KPa), all above the threshold condition. The selected scale is such that the displacement curves for 5 psi and 10 psi appear to be straight lines, but when magnified 100 times (lower curves) they reveal the expected non-linearity of cantilever bending. It should also be noted that for loads much beyond the threshold, a doubly curved shape with an inflexion point near the major load application is expected. This trend shows up clearly in the 60 psi (414 KPa) curve with inflexion point near Node 49.

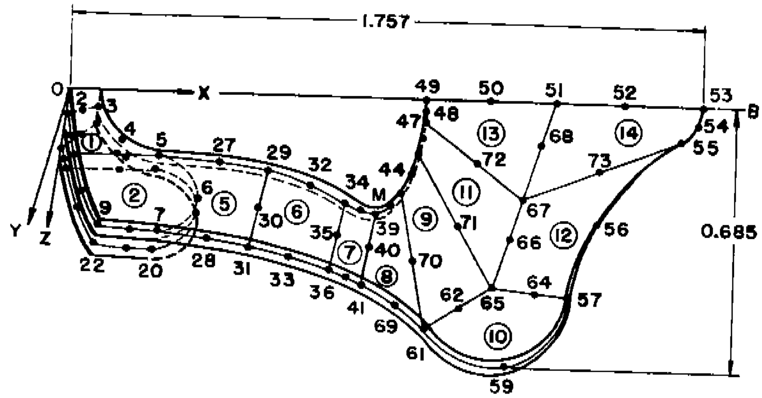

Figure 2. Finite element model of the valve using the SUPERB program.

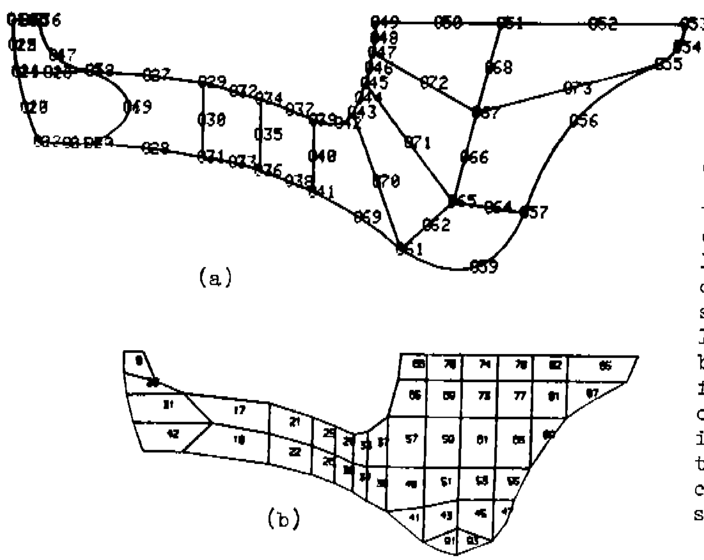

Figure 3. Computer plotted model geometry, (a) from the SUPERB program, and (b) from the ANSYS program.

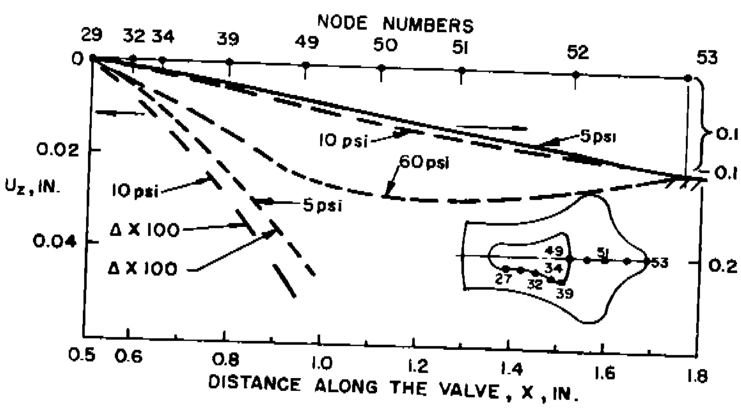

Figure 4. Hand plots of z displacements, U_z, along the length of the valve.

The U_z displacement of a particular location likely to show appreciable change after valve tip stoppage can also be plotted as a function of the applied load. Figure 5 is such a plot with z displacement of Node 51. It is seen that although the relationship is linear both before and after the threshold load, the slope changes. This change is expected because of a shift in the mechanics of deformation from a cantilever type beam to one that is fixed at one end and simply supported at the other. It is interesting to note that by plotting this way the threshold pressure, 2.3 psi (15.87 KPa) in this case, can be determined easily and accurately by the intersection point of the two straight lines.

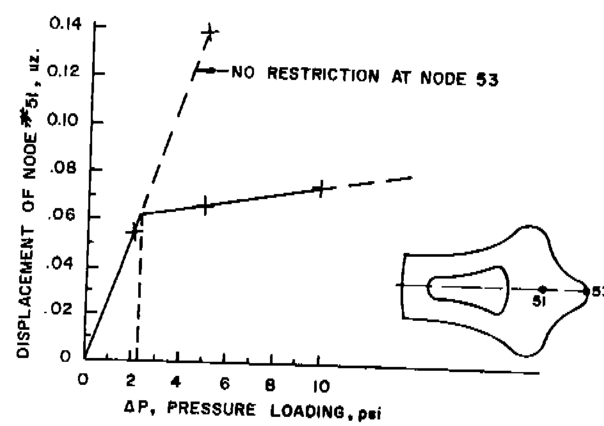

Figure 5. The z displacement, U_z, of Node 51 as a function of the pressure loading.

The stress solution revealed the peak value of the maximum principal stress to occur at Node 29 for all the load levels applied. This information assures the designer that in the static mode of deformation, the stress concentration due to the deep cutout in Region M, Figure 2, is not critical. Figure 6 shows the variation of the maximum stress at Node 29 as a function of the pressure load, ΔP, which is very similar to that of the displacement, Figure 5. The slope change occurs at exactly the same threshold pressure of ΔP = 2.3 psi (15.87 KPa), as in the previous case.

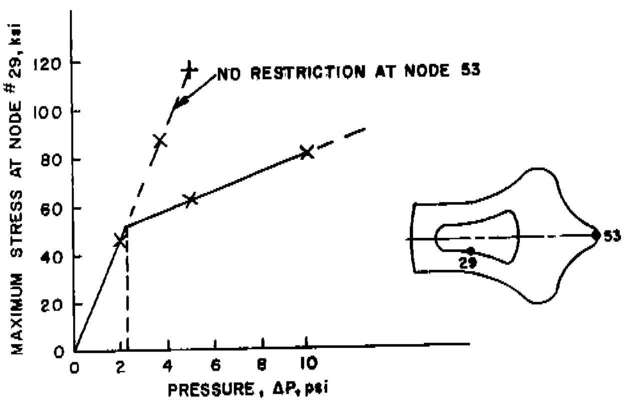

Figure 6. The maximum principal stress at Node 29 as a function of the pressure loading.

The combination of the actual valve load and the valve design (geometry and material) should be such that the maximum stress remains below the fatigue limit of the valve material. The static or fundamental mode analysis is not enough to assure fail safe design or indicate performance characteristics of the valve. Some higher modes of natural vibration, when excited, may shift the critical deformation area to another region. In addition, the natural frequencies should be known to allow appropriate correlation of the valve and crankshaft motions for producing optimum capacity and efficiency of the compressor. A dynamic analysis is therefore essential.

DYNAMIC ANALYSIS

The equations of motion in matrix form for free and undamped vibration of a finite element model of a structure are given by, (1,5)

$$[K]\{\delta\} + [M]\{\ddot{\delta}\} = 0 \qquad (1)$$

where $[K]$ = the system stiffness matrix

$\{\delta\}$ = vector of nodal displacements

$[M]$ = system mass matrix, and

$\{\ddot{\delta}\} = \frac{\partial^2}{\partial t^2}\{\delta\}$

The system stiffness matrix [K] is the same as assembled for static analysis. The only new parameter needed is the mass matrix [M]. This is obtained by appropriate summation of the elemental mass matrices, $[m_{ij}]$ of the model,

$$[M] = \sum_{\substack{i=1\\j=1}}^{n} [m_{ij}] = \sum_{1}^{n} \int_{V} [N_i]^T \rho [N_j] dV \qquad (2)$$

where $[N_i]^T$ and $[N_j]$ are the same "shape functions" as used during the stiffness matrix formulation.

The T stands for transpose of the matrix. ρ is the mass density of the material, and V indicates integration over the elemental volume. Thus ρ is a new material property input for dynamic analysis.

If it is assumed that the vibrations are harmonic of the form,

$$\{\delta\} = \{\delta_o\} \cos \omega t$$

where ω is the angular frequency, and t is the time, then Equation (2) transforms into a typical eigenvalue problem,

$$\left[-\omega^2 [M] + [K]\right] \{\delta_o\} = 0. \qquad (3)$$

The solution of these equations gives a set of natural frequencies, ω_n, and corresponding vibrational modes $\{\delta_o\}_n$. Here n equals the total number of unconstrained degrees of freedom.

After reduction to standard eigenvalue form, Equation (3) is solved by the Jacobi iteration method. Use is also made of matrix condensation or the Guyan Reduction technique in which a large structural system is characterized by only a small set of master or dynamic degrees of freedom. The mode shape solutions can then be expanded by interpolation to the full structural set, and plotted thereafter if desired. Both SUPERB (4) and ANSYS (6) utilize these techniques in their solution process. In SUPERB, however, the set of dynamic degrees of freedom can either be selected and input directly by the user, or selected automatically by the program. Since both mass and stiffness matrices have to be stored, a dynamic problem can be only about 70% as large (in terms of the total active degrees of freedom) as an equivalent static problem.

SUPERB Modeling

The modeling and input of a modal analysis problem with SUPERB follows the same general pattern as the static analysis. If the extraction of only symmetric modes is sufficient in a given problem, then the advantage of symmetry conditions in the modeling may be retained, and the same mesh geometry can be used. In the present problem, the finite element model was the same as shown in Figure 2. The displacements in the z direction, U_z, of selected nodes along the valve length comprised the input of 28 dynamic degrees of freedom. The natural frequencies and shapes of the first five symmetric modes were requested as the output.

ANSYS Modeling

Since SUPERB dynamic capability with isoparametric elements is a very recent release and may have some unrecognized problems, it was decided to perform a parallel analysis using the ANSYS program (with the flat conventional shell element) which has been tested and verified for accuracy for a longer period of time. This time the full valve geometry was used, Figure 7, to extract all the possible modes. It should be noted that there are 94 STIF 63* elements used with an equivalent of 47 elements for half the model as shown beside the SUPERB model in Figure 3, for comparison. Note that still the valve boundaries are not as accurately represented as in the SUPERB model with only 14 elements. This comparison clearly demonstrates the superiority of isoparametric elements over conventional elements in geometrical modeling. There were 78 dynamic degrees of freedom (with an equivalent of 39 for half the model) selected.

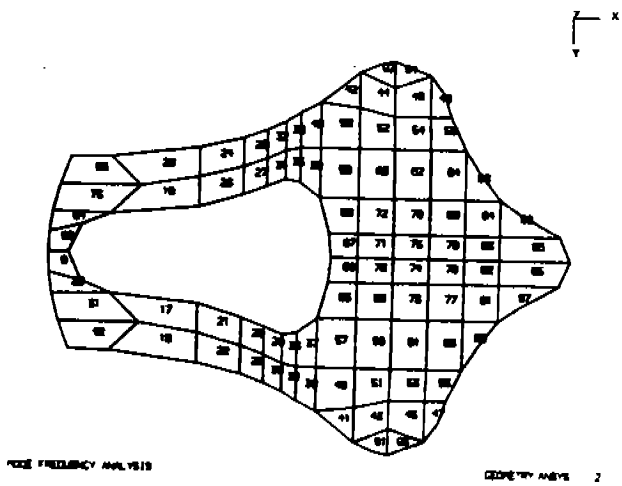

Figure 7. Computer geometry plot of the FEM model using the ANSYS program.

Dynamic Results

The results obtained for the first three modes are summarized in Table I for both SUPERB and ANSYS solutions. The comparison shows that the natural frequency from the SUPERB model, 228 hz, is about twice that from the ANSYS model, 110 hz. It should be recognized that the modes and frequencies above the first may not correspond with each other due to the omission of some non-symmetric modes in the SUPERB model, but the fundamental mode should be the same for both. The computer plots of the first mode shape from both the SUPERB and ANSYS models represented the cantilever type bending accurately.

*STIF is a computer code name in the ANSYS finite element library to identify the available element formulations.

TABLE I

MODE	FREQUENCY, hz	
	SUPERB	ANSYS
1	228.48	109.56
2	875.2	596.57
3	1825.7	1084.2

DISCUSSION

From simple experiments performed it appears that the SUPERB dynamic program does not give very accurate results. It has been realized recently that a relatively coarser mesh that may prove to be sufficient for accurate static analysis, may be too stiff for dynamic response, and may give much higher frequencies than the true values (7). In our case, the ANSYS model (47 elements and 39 dynamic degrees of freedom) is definitely a finer mesh compared to the SUPERB model (14 elements and 28 d.d.o.f.), but the difference in answers is too large. The requirement of a finer mesh in SUPERB will necessitate extra work in the mesh structure already created for static analysis, and force at least a partial loss of the advantages derived from isoparametric formulation.

SUPERB is strictly a linear program but it is adequate for performing stress analysis of valves because the stresses should always remain within the elastic range. In fact, the maximum stress calculated should not exceed the fatigue limit of the material.

The dynamic solution may help in evaluating the volumetric performance of a compressor valve. For instance, consider the suction valve analyzed in this study for use in a 1800 rpm compressor. The first mode vibrational frequency of the valve from the ANSYS solution is 110 Hz which is approximately four times that of the crank motion. This will lead to approximately two cycles of valve opening and closing instead of one during the suction half cycle of the crank, as illustrated schematically in Figure 8. The valve is closed at the 90° position when it is supposed to be wide open. This will result in poor volumetric efficiency of the compressor. This example clearly demonstrates the value of the dynamic analysis to the valve designer.

It should be noted that in many cases antisymmetric modes of vibration can lead to critical stressing of the valve and therefore these modes should not be ignored when considering the dynamic analysis. The solution obtained from the ANSYS model in our case, includes all the possible modes, symmetric and antisymmetric, and is more meaningful and valuable to the designer.

Experience had shown that the type of valve analyzed is susceptible to fail near Node 39 (location M), Figure 2. No such indication is obtained from the static analysis which shows the peak stress near Node 29. However, if we examine the third vibrational mode (second cantilever mode) from the ANSYS solution shown in Figure 9, we see a node (zero displacement) N occurring slightly towards the right of the critical location M. The flapping of the valve with respect to N may lead to breaking stresses in that area. The excitation of this damaging third mode will be accentuated by the valve tip contact with its stop.

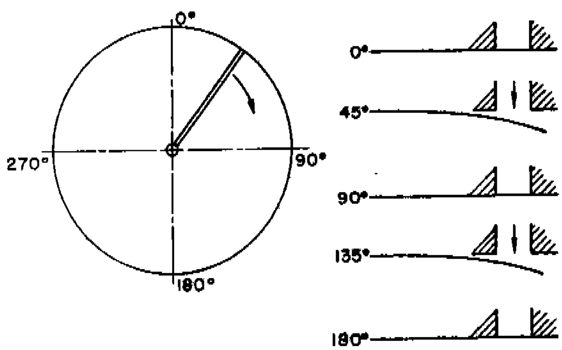

Figure 8. Correspondence of crank motion with the valve motion when mode 1 of the valve is excited (at 1800).

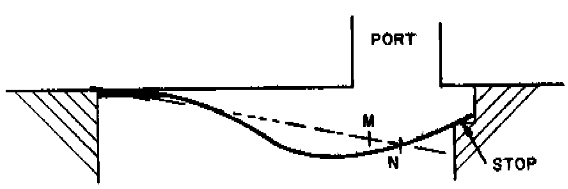

Figure 9. The third mode of natural vibration of the valve. ANSYS solution.

CONCLUSIONS

(1) With the example of a typical cantilever type reed valve, it was demonstrated that the finite element method in general, and easily accessible SUPERB program in particular, is ideally suited for static stress analysis of compressor valves.

(2) The static analysis alone is insufficient in describing the valve behavior completely because the excitation of many dynamic characteristics is possible during the compressor operation. Therefore both the static and dynamic analyses are required.

(3) The dynamic analysis with SUPERB may need a finer mesh than the static analysis for accurate results.

(4) The experience and knowledge of the actual valve function will aid in understanding the dynamic analysis.

(5) The Finite Element Method may appear expensive and time consuming in the beginning, but the routine use may prove to be the most efficient and economical method available for the valve stress analysis.

REFERENCES

1. O.C. Zienkiewicz, The Finite Element Method in Engineering Science, McGraw-Hill Book Co., London, 1971.

2. R.H. Gallagher, Finite Element Analysis: Fundamentals, Prentice-Hall, Inc., New Jersey, 1975.

3. J.R. Friley and J.F. Hamilton, "Characterization of Reed Type Compressor Valves by the Finite Element Method", Proc. 1976 Purdue Compressor Tech. Conf., July 6-9, 1976, p. 295.

4. Superb User's Manual, Version 5.0, March 1978, issued by Structural Dynamics Research Corporation (SDRC), 5729 Dragon Way, Cincinnati, Ohio, 45227.

5. K.H. Huebner, The Finite Element Method for Engineers, John Wiley and Sons, New York, 1975.

6. Ansys User's Manual, Version 2.0, 1975, issued by Swanson Analysis Systems, Inc., Elizabeth, Pennsylvania 15037.

7. Steven Weiner, SDRC, Cincinnati, Ohio, private communication, March 1978.