USING MSC/NASTRAN TO OBTAIN MODAL PARAMETERS

Cuann-yeu Chang Yuan-bing Chang

Aeronautical Research Laboratory, AIDC, CSIST
P. O. Box 8676-1-3, Taichung, Taiwan 40722,
Republic of China.

ABSTRACT:

Natural frequencies and mode shapes are fundamental parameters for almost all dynamic analyses. It is not difficult to obtain a set of accurate results if users are familiar with finite element characteristics and NASTRAN usages. Herein the whole pictures are dealt with and the discussions on each procedure are made for performing the modal analysis of air vehicles. First of all, guidelines to determine how fine the mesh is for different modeling objectives are studied. At the same time, the capability of using suitable finite elements to reflect real characteristics of the global and local system is essential and worth to note. On the other hand, for a large structure, not all the d.o.f.'s can be put into eigensolver, only a small set can be left, the degree of accuracy remained have to be considered prudently. Thus, the general rule of dynamic reductions provided by NASTRAN is discussed. Meanwhile, several error messages usually occurred are noticed and the keys to avoid them are hereby mentioned, too. Finally the comparison between analytical results and testing results is also conducted by NASTRAN DMAP and some externally written FORTRAN programs.

1. Introduction

Natural frequencies and mode shapes are fundamental parameters for almost all dynamic structural systems. These modal data are very useful and important because they can be treated as a set of orthogonal generalized coordinates which will be implemented effectively in dynamic analyses by using modal approach.

To obtain the modal parameters, it is necessary to establish a mathematical model first. And this model has to represent faithfully the real characteristics of the actual system. Then, by using finite element method and appropriate structural matrix operations, the stiffness and mass matrices of entire discretized system can be obtained. After the condensation process is implemented, a real eigenvalue problem can be solved by numerical procedure.

In this study, a series of practical procedures to extract modal parameters are discussed, which are environmental effects, mesh size determination, finite element modeling techniques, dynamic reduction techniques, error analysis, and correlation between analytical and test results. And in the end, the conclusion is made.

2. Environmental Conditions

Before performing the structural analysis, it is suggested to measure the environmental conditions. This will give us a distinct guideline to implement suitable normal modes for the dynamic analyses in modal approach.

The dynamic loads have to be obtained by operational (flight) load test, then the energy distribution in different frequency range will be known by the process of test data reduction and analysis. Because any structural system with its own characteristic transfer function will have different response sensitivity on those input forcing functions of different energy distribution in frequency domain, all the modes resided in the interested frequency range should be considered. For aircraft, a simple and conservative way to consider the environmental effect is to include all modes below 100 HZ in analysis. However, this frequency limit is to be tuned due to engineering judgement.

3. Mesh Size Determination

To establish an appropriate finite element model, it is worth to set up a criterion, deciding how fine the mesh should be and how many grids (d.o.f.'s) should be used.

For the aircraft dynamic modeling, the lower and upper bounds of finite element dimension are existed. Practically, it is not allowed to use too many elements for the demand to improve the analytical accuracy. Generally, a economic rule will provide the lower limit value of element size. It is suggested to perform a sensitivity analysis of convergence rate (accuracy achieved/no. of d.o.f. used) vs. resources (computer & human being). A conclusive trade-off is recommended.

It is especially important to perform aeroelastic analyses for flight vehicles. The aerodynamic force can be integrated over a set of collocation points in an aerodynamic mesh. This mesh is quite different from the structure mesh. Some kind of transformation between these two meshes must be implemented to provide a linkage and perform the aeroelastic analysis. Generally, the algorithm of spline is recommended. During the interpolation process of spline, a determination of minimum grid points to be used along each direction for all usable modes should be made. For example, if we have to deal with the third bending mode (Ref. Fig. 1) of a control surface in the interested frequency range, i.e., 4 nodal points without displacement along the hinge line, then 21 grids along this direction is requested assuming 5 grids can represent a smooth curve. Thus, it is concluded that the upper limit of element size is available by estimating the required accuracy of mesh transformation, while the lower bound is determined by resource limitation.

4. Modeling Technique

The finite element method has become the universally adopted procedure to perform the structural analysis. The choice of finite element type to simulate the actual structure behaviour is fundamentally in need of engineering experience. To obtain valid results, the user sometimes may make a reasonable adjustment in stiffness by local reinforcement to match the correct load path. But, generally speaking, they have to try their best to correlate the mathematical response of the finite element model to the actual behavior. This is the art of structural dynamics coming into play.

The following elements in MSC/NASTRAN finite element library are recommended to be used in modeling airvehicle after the force equilibrium and deformation compatibility on boundaries and element interfaces are well satisfied.

1) CONROD - an axial and torsional member: beam caps, longerons, stringers and stiffeners.
2) CBAR (CBEAM) - a bending, torsion and axial member: longerons, beams and frames.
3) CSHEAR - a shear panel: buckled or unbuckled skin and webs where shear behaviour alone is important.
4) CQUAD4 - a flat 4 node isoparametric panel, carries shear, normal, transverse and bending loads: skins, webs, sandwich and composites. This is the recommended panel element for most modeling.
5) CQUAD8 - an eight node flat or curved isoparametric shell element: same uses as CQUAD4.
6) CTRIA3 - a three node flat panel element. It is a companion to the CQUAD4 four node element. However, the in-plane stress is constant for this element.
7) CTRIA6 - a six node triangular isoparametric curved element that complements the CQUAD8.
8) CHEXA - a solid isoparametric eight to twenty node element: for modeling of solids.
9) CPENTA - a solid isoparametric six to fifteen node solid element companion to the CHEXA.

In addition to the element selection, there are many other aspects should be noticed, such as the effect of grid sequencing on bandwidth, using low aspect ratio panels, good compatibility along a long distance element boundary, boundary conditions and rigid body constraints.

Structural modeling technique is mixed by science and art. Good engineering judgement is probably the most important asset which comes from the user's experience and if the user is well understanding the structural behaviour. Except the understanding of the finite element theory, the ability to correlate the mathematical behaviour of the finite elements in the program library to the behaviour of the real structural elements is needed for realistic structural idealization and valid interpretation of the results. In summary, the results of any finite element analysis are only as good as the model used and the way the output is interpreted.

5. Dynamic Reduction

A dynamic analysis can be separated into three phases: assembly of dynamic equations, solution of dynamic equations and data recovery of response quantities such as forces and stresses. As problem size increases, the cost of the second phase is dominant. [1] It increases as the square or the cube of the number of d.o.f.'s, whereas others are linear.

There are two different dynamic reductions, Guyan Reduction and Generalized Dynamic Reduction, provided by MSC/NASTRAN. The details are described in Ref. 2.

For aircraft lifting surface, such as wing, horizontal tail and vertical fin, it is recommended to use Guyan Reduction, because it is cheap and easy to handle. You may just uniformly distribute the a-set d.o.f.'s, and pick up suitable points for downstream spline's use. The guidelines due to the author's experience are outlined as follows:

1) Select points with large lumped mass
2) Distribute all a-set points uniformly over the structure
3) Select points used in spline

6. Error Detection and Diagnosis

6.1 Automatic Output Message

The messages which are automatically generated by MSC/NASTRAN [2,3] in printout of modal analysis are outlined as follows:

1) User information message 3035

   The third column of Table 1 represents the strain energy resulting from moving the structure through a unit displacement at listed support d.o.f. with the other support points locked. For a good choice of support point, the strain energy printed in this message is caused by the MPC card, coordinate transformation, large local stiffness ratios and the other modeling complexities.

2) VAXW table

   Table 2 appears when the auto-omit operation takes place. It lists the external sequence number of all points omitted. This table should be reviewed during model development to ensure that all d.o.f.'s listed are massless by intent and not by oversight.

3) User warning message 4698

   When MSC/NASTRAN detects the model may have mumerical ill-conditioning problems, it starts issuing a series of related messages, such as user information 4158 and some other unnumbered messages.

   This warning message 4698 (Ref. Tab. 3) lists all external sequence numbers for points that have symmetric decomposition factor to stiffness matrix diagonal term ratios larger than the one defined by "Param Maxratio" in bulk data deck. The default value of Maxratio is currently 1.0E5.

   A well conditioned problem should have this ratio between 10^0 - 10^2. Large values of this ratios indicate some degree of ill-conditioning.

Theoretically, rigid body motion should result in an infinite value for this ratio, however, computer round off will result in a small but nonzero value on the diagonal for each possible rigid body motion that exists in the model.

On the other hand, a well formed problem should have no negative term on the factor diagonal. Negative terms on the diagonal of the triangular factor would indicate that the original matrix was not positively definite.

4) Grid Point Singularity Table (GPST)

   Table 4 will be printed out if local grid point singularities exists whether or not automatic SPC generation is requested.

   In general, the user should carefully review the GPST before accepting the results of an analysis, then, they are able to see if there exist any undesirable constraints.

6.2 User Requested Diagnosis

Sometimes users should request some special output by themselves for detecting implicit errors.

1) Diagnosis requested by DIAG cards in the executive control deck.

   DIAG 8 - print matrix trailers

   DIAG 16 - inverse power diagnosis

   DIAG 23 - element strains

2) User selected exclusive set in given format through the request by "PARAM USETPRT" and "PARAM USETSEL" in bulk data deck.

3) Weight and C.G. verification by comparing against output of "PARAM GRDPNT" in bulk data deck.

Sol. 3 will print "OUTPUT FROM GRID POINT WEIGHT GENERATOR", through DMAP no. 73 module GPWG, indicating the mass, mass moment of inertia and C.G. position. Before having checked this information, modal analysis results should not be used.

4) Element Strain Energy request via "ESE = ALL" in case control deck. For each mode, it is recommended to check out the strain energy distribution and see if there exists any abnormal or unreasonable amount of energy absorbed by some element.

5) Single Point Constraints Force request via "SPECFORCE = ALL" in case control deck.

   For some improper constraints in one model, the reaction force requested should be checked. If local singularities are removed by AUTOSPC, the reaction force at these d.o.f.'s need to be checked and see if it is computed zero or not.

6) Normalization check

   For each mode, a check should be performed after data recovery to see if there is any d.o.f. whose corresponding value in each mode vector is greater than 1.0. Usually these values after normalization process, greater than 1.0 are caused by improper ASET selection or defective modeling set-up.

6.3 Final Diagnosis

After the whole procedures described above have been executed completely, still there is an another way to verify the modal analysis results, i.e., by means of similarity check. That means, by finding a similar model which have been worked out and the report is available and then comparing your results with those described in report. The comparison should be correlated closely for several obviously identical modes.

This is a very valuable approach to be known in design phase since there is no structural test data existed.

7. Correlation Between Analytical and Test Results

Modal tests are playing an increasingly important role in structural dynamic efforts which are in needs of analytical model verification and trouble shootings. Moreover, the adjustment (tuning) of analytical finite element model is especially important for project success, and it has to be performed fully in accordance with modal testing results.

If no error or warning messages, which were discussed in section 6, founded in MSC/NASTRAN printout and a similarity check have been passed, we can conclude the modal parameters of our analytical model is quite credible. In this situation, we may proceed to perform the comparisons between analytical and testing results by internally implementing DMAP operations and/or externally by self-developed FORTRAN programs without difficulties.

Except those described in Ref. 5, a simple correlation criterion is recommended and expressed as follows:

$$\text{abs}\,([\phi]^T[\hat{\phi}] - [I]) \leq [\varepsilon]$$

where $[\phi]_{mxn}$ is normalized test mode shapes

$[\hat{\phi}]_{mxn}$ is normalized analytical mode shapes

$[1]_{nxn}$ is identity matrix

$[\varepsilon]_{nxn}$ is epsilon, determined by how close the correlation is requested.

m is no. of d.o.f's

n is no. of modes used

8. Concluding Remarks and Recommendations

It is worth to note that the efforts spent in the creation of an accurate structural model can be rewarded by a substantial reduction of the modal testing extents. Therefore, simulating the actual structure characteristics and using the modal testing results to tune the analytical model by applying perturbation theory is a distinct regular route. It is the author's belief that these usage mentioned in the present study will be matured further and become the standard practice.

On the other hand, the checkpoint and restart features of MSC/NASTRAN are very useful and effective especially for extracting modal parameters of aircraft with different fuel and external store configurations. Meanwhile, DMAP and its corresponding internal parameters for controlling program modules operation are recommended to be used. They will lead us to be an efficient structure dynamic analyst.

9. References

[1] Craig, Roy R., Jr., "Structural Dynamics - An Introduction to Computer Method," John Wiley & sons, Inc., New York, 1981.

[2] Gockel, M.A., Editor, "MSC/NASTRAN Handbook for Dynamic Analysis, Version 63," The MacNeal-Schwendler Corporation, June 1983.

[3] McCormick, Caleb W., Editor, "MSC/NASTRAN User's Manual, Version 63," The MacNeal-Schwendler Corporation, May 1983.

[4] McLean, Donald M., Editor, "MSC/NASTRAN Programmer's Manual, Version 63," The MacNeal-Schwendler Corporation, Oct. 1983.

[5] Chen, Jay-Chung, "Evaluation of Modal Testing Methods," AIAA 84-1071.

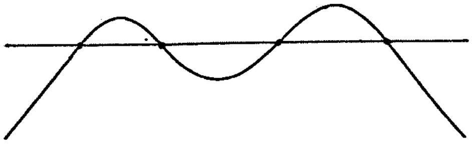

Fig. 1 Illustration for third bending mode

TABLE 1. User information message 3035

```
*** USER INFORMATION MESSAGE 3035 FOR DATA BLOCK KLR
SUPPORT PT. NO.   EPSILON       STRAIN ENERGY   EPSILONS LARGER THAN 0.001
                                                ARE FLAGGED WITH ASTERISKS.

            1  1.0584467E-05   1.1970776E+00

            2  1.0584467E-05   0.0000000E+00

            3  1.0584467E-05   1.1970776E+00
```

TABLE 2. VAXW table

```
          POINT       VALUE       POINT       VALUE       POINT       VALUE
COLUMN
          113  S   1.00000E+00    114  S   1.00000E+00    115  S   1.00000E+00
          118  S   1.00000E+00    119  S   1.00000E+00    120  S   1.00000E+00
```

TABLE 3. User warning message 4698

```
*** USER WARNING MESSAGE --- ROW 7 of LOW. TRI. FACTOR HAS DIAGONAL
    TERM = 0 (OR .LT. 0 IF CHOLESKY)

*** USER INFORMATION MESSAGE --- 1.0E-10 SUBSTITUTED FOR DIAG.  TERM
    OF LOW. TRI. FACTOR AT ROW   7

*** USER INFORMATION  MESSAGE 4158 --- STATISTICS FOR SYMMETRIC DECOM-
    POSITION OF DATA BLOCK KOO  FOLLOW MAXIMUM RATIO OF MATRIX DIAGONAL
    TO FACTOR DIAGONAL = 1.9E+16  AT ROW NUMBER  7

*** USER WARNING MESSAGE 4698.  STATISTICS FOR DECOMPOSITION OF MATRIX
    KOO THE FOLLOWING DEGREES OF FREEDOM HAVE FACTOR DIAGONAL RATIOS
    GREATER THAN 1.00000E+05 OR HAVE NEGATIVE TERMS ON THE FACTOR  DIAGONAL.

GRID POINT ID    DEGREE OF FREEDOM    FACTOR DIAGONAL RATIO    MATRIX DIAGONAL
    10006                T1                1.88496E+16            1.88496E+06
```

TABLE 4 Grid point singularity table

```
POINT     TYPE       FAILED       STIFFNESS      OLD       NEW
 ID                 DIRECTION       RATIO        USET      USET


 211       G           6             0.            0         S      *
 212       G           4             0.            0         S      *
 214       G           4             0.            0         S      *
 215       G           4             0.            0         S      *
```