NAVAL POSTGRADUATE SCHOOL
MONTEREY, CALIFORNIA

THESIS

SUBMARINE MACHINERY CRADLE:
STRUCTURAL DYNAMIC DESIGN AND ANALYSIS TECHNIQUES
USING FREQUENCY DOMAIN STRUCTURAL SYNTHESIS

by

Ronald E. Cook

March, 1994

Thesis Advisor: Joshua H. Gordis

Approved for public release; distribution is unlimited.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1a. REPORT SECURITY CLASSIFICATION: Unclassified	1b. RESTRICTIVE MARKINGS
2a. SECURITY CLASSIFICATION AUTHORITY	3. DISTRIBUTION/AVAILABILITY OF REPORT: Approved for public release; distribution is unlimited
2b. DECLASSIFICATION/DOWNGRADING SCHEDULE	
4. PERFORMING ORGANIZATION REPORT NUMBER(S)	5. MONITORING ORGANIZATION REPORT NUMBER(S)
6a. NAME OF PERFORMING ORGANIZATION: Naval Postgraduate School; 6b. OFFICE SYMBOL (If applicable): ME	7a. NAME OF MONITORING ORGANIZATION: Naval Postgraduate School
6c. ADDRESS (City, State, and ZIP Code): Monterey, CA 93943-5000	7b. ADDRESS (City, State, and ZIP Code): Monterey, CA 93943-5000
8a. NAME OF FUNDING/SPONSORING ORGANIZATION; 8b. OFFICE SYMBOL (If applicable)	9. PROCUREMENT INSTRUMENT IDENTIFICATION NUMBER
8c. ADDRESS (City, State, and ZIP Code)	10. SOURCE OF FUNDING NUMBERS: PROGRAM ELEMENT NO; PROJECT NO.; TASK NO; WORK UNIT ACCESSION NO

11. TITLE (Include Security Classification)
SUBMARINE MACHINERY CRADLE: STRUCTURAL DYNAMIC DESIGN AND ANALYSIS TECHNIQUES USING FREQUENCY DOMAIN STRUCTURAL SYNTHESIS

12. PERSONAL AUTHOR(S)
Ronald Edward Cook

13a. TYPE OF REPORT	13b. TIME COVERED	14. DATE OF REPORT (Year,Month,Day)	15. PAGE COUNT
Master's Thesis	FROM _____ TO _____	March 1994	188

16. SUPPLEMENTARY NOTATION
The views expressed in this thesis are those of the author and do not reflect the official policy or position of the Department of Defense or the U.S. Government.

17. COSATI CODES			18. SUBJECT TERMS (Continue on reverse if necessary and identify by block number)
FIELD	GROUP	SUB-GROUP	Frequency Domain Structural Synthesis

19. ABSTRACT (Continue on reverse if necessary and identify by block number)

The tactical implications of submarine acoustic radiation and UNDEX-survivability have motivated the development of an advanced machinery cradle which will provide shock and vibration isolation of the submarine internals, thereby minimizing the resulting acoustic radiation. The cradle space frame must be designed and optimized for both minimum shock/vibration bi-directional transmissibility and minimum total cradle weight. Frequency domain structural synthesis (structural modification and substructure coupling), is applied to the cradle design. The method addresses static and complex dynamic problems in structural design analysis, and allows the direct analytic treatment of specialized equipment, such as frequency-dependent visco-elastic isolators.

20. DISTRIBUTION/AVAILABILITY OF ABSTRACT	21. ABSTRACT SECURITY CLASSIFICATION	
[X] UNCLASSIFIED/UNLIMITED [ ] SAME AS RPT. [ ] DTIC USERS	Unclassified	
22a. NAME OF RESPONSIBLE INDIVIDUAL: Joshua H. Gordis	22b. TELEPHONE (Include Area Code): (408) 656 - 2866	22c. OFFICE SYMBOL: ME/Go

Approved for public release: distribution is unlimited

Submarine Machinery Cradle: Structural Dynamic Design and Analysis Techniques Using Frequency Domain Structural Synthesis

by

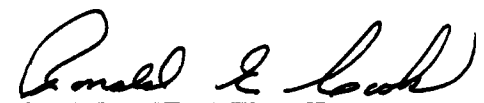
Ronald E. Cook
Lieutenant, United States Navy
B.S.P.E., University of Wyoming, 1985

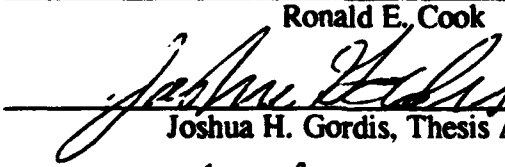
Submitted in partial fulfillment of
requirements for the degree of

MASTER OF SCIENCE IN MECHANICAL ENGINEERING

from the

NAVAL POSTGRADUATE SCHOOL
March 1994

Author: 
Ronald E. Cook

Approved by: 
Joshua H. Gordis, Thesis Advisor


Matthew D. Kelleher, Chairman,
Department of Mechanical Engineering

ABSTRACT

The tactical implications of submarine acoustic radiation and UNDEX-survivability have motivated the development of an advanced machinery cradle which will provide shock and vibration isolation of the submarine internals, thereby minimizing the resulting acoustic radiation. The cradle space frame must be designed and optimized for both minimum shock/vibration bi-directional transmissibility and minimum total cradle weight. Frequency domain structural synthesis (structural modification and substructure coupling), is applied to the cradle design. The method addresses static and complex dynamic problems in structural design analysis, and allows the direct analytic treatment of specialized equipment, such as frequency-dependent visco-elastic isolators.

TABLE OF CONTENTS

I. INTRODUCTION 1

II. FINITE ELEMENT FORMULATION 3

III. FREQUENCY DOMAIN STRUCTURAL SYNTHESIS 16

A. GENERALIZED FREQUENCY RESPONSE 18

B. MATRIX PARTITIONING 23

C. STRUCTURAL MODIFICATION AND INDIRECT SUBSTRUCTURE COUPLING 26

D SUBSTRUCTURE COUPLING AND CONSTRAINT IMPOSITION 31

E. DIRECTED GRAPHS AND MAPPING MATRICES 36

F. MODIFICATION AND INDIRECT COUPLING USING MAPPING MATRICES 39

IV. NUMERICAL EXAMPLES 44

A. EXAMPLE (1): DYNAMIC INDIRECT COUPLING 45

B. EXAMPLE (2): DYNAMIC DIRECT COUPLING 49

C. EXAMPLE (3): STRUCTURAL MODIFICATION (REMOVAL OF A BEAM ELEMENT) 54

D. EXAMPLE (4): STRUCTURAL MODIFICATION (ADDITION OF A BEAM) 59

E. EXAMPLE (5): INDIRECT COUPLING WITH ISOLATORS 62

F. EXAMPLE (6): INDIRECT COUPLING WITH FREQUENCY DEPENDENT ISOLATORS 68

G. EXAMPLE (7): STRESS CALCULATION BY DYNAMIC INDIRECT COUPLING 72

H. EXAMPLE (8): DYNAMIC DIRECT COUPLING USING MODAL REPRESENTATION OF FRF 80

V. CONCLUSIONS / RECOMMENDATIONS 90

APPENDIX A MATLAB CODE FOR EXAMPLE ONE 92

APPENDIX B MATLAB CODE FOR EXAMPLE TWO 100

APPENDIX C MATLAB CODE FOR EXAMPLE THREE 111

APPENDIX D MATLAB CODE FOR EXAMPLE FOUR 121

APPENDIX E MATLAB CODE FOR EXAMPLE FIVE 130

APPENDIX F MATLAB CODE FOR EXAMPLE SIX 141

APPENDIX G MATLAB CODE FOR EXAMPLE SEVEN 151

APPENDIX H MATLAB CODE FOR EXAMPLE EIGHT 158

APPENDIX I GENERAL MATLAB FUNCTIONS 174

LIST OF REFERENCES 180

INITIAL DISTRIBUTION LIST 181

I. INTRODUCTION

The design of complex structural dynamic systems requires the building of detailed mathematical models with which to predict static and dynamic response. Most commonly, the finite element (FE) method is used to generate structure system matrices with which dynamic response can be calculated. While the FE method currently provides the best means of predicting response for complex structural systems, the time required to assemble the system matrices and to process them for the calculation of dynamic response can be prohibitive. Therefore, the use of the FE method for performing design analyses often precludes the performance of numerous design analyses in the search for an optimal design. This is especially true when a FE based analysis is to be used in conjunction with advanced design techniques such as optimization. The iterative process of modeling the system and analyzing the model to determine the system performance is the design-analysis cycle.

The traditional design-analysis cycle consists of the following process. A designer builds a FE model which best represents the system. The system model is the complete system structure, for example, a submarine hull and an internal machinery support cradle is modeled as one structure. The definition of the FE model yields system matrices, which include stiffness, mass, and less commonly damping. The numerical generation of the system matrices is referred to as the assembly phase. At this point, loads are applied and responses, static and/or dynamic, are calculated. The calculation of system response is referred to as the solution phase. The responses are then used to calculate stresses and strains in the model. These calculations are referred to as the post-processing phase. The solution phase is the most costly in terms of time and computing resources.

Based on the acceptability of the displacements, stresses, and strains calculated, the design or system model may have to be changed in the interest of improving the response characteristics of the initial or follow on design. For example, a high stress which is unacceptable may exist at a certain location in the design. The designer decides that if a particular alteration is made to the design, the stress response will become within tolerable levels. Traditionally, this alteration requires a repeat of the assembly, solution, and post-processing phase of the analysis, a cost and time intensive procedure which limits the number of design re-analyses that can be accomplished. Since the re-analysis is time consuming, the optimal design is abandoned for a final design which is less than optimal.

Therefore, with the intent of accelerating the design process and lowering the attendant costs, new methodologies for assembling and modifying system models is presented. The new method replaces all three of the FE analysis phases with a single computationally efficient calculation. The method to be described herein, generally referred to as frequency domain structural synthesis [Refs. 1,2,3], is directed specifically at drastically reducing the time required to perform a design analysis cycle. This capability for rapid re-analysis makes structural synthesis ideal for use in advanced automated design environments, such as in conjunction with optimization codes.

II. FINITE ELEMENT FORMULATION

The finite element method used for comparison with the solution obtained from frequency domain structural synthesis is based on Lagrange's equation of motion [Ref. 4]. Various types of elements are used in modeling of structural systems, including for example, plate, shell, and beam elements. Our discussion will be limited to beam elements experiencing combined bending and axial deformations. We are using beam elements to demonstrate the methodology because the beam element allows for a manageable system of equations and matrices that are easily handled by a personal computer. The theory remains valid for all types of elements and is unaffected by the complexity of shell or plate elements. Beam elements that are subject to bending and axial deformation have three joint displacements at each end of the beam element. The beam element has six generalized coordinates and six degrees of freedom (DOF), which yields a mass, stiffness, and damping matrix for the beam element of size (6 x 6). The beam element is shown in Figure 1.

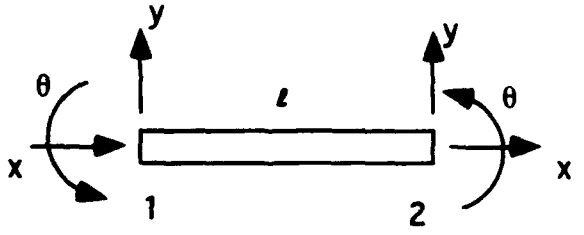


Figure 1. Beam Element with Coordinate and Nodal Orientation

Each node has a set of coordinates, axial (x), lateral (y), and rotational (θ), however elements are not limited to three DOF. Elements can be modeled with six DOF per node.

The derivation presented in reference[4] assumes that the axial forces associated with the axial joint displacements (x) have only a negligible effect on the shape functions associated with the joint displacements (y) and (θ). With this assumption the mass and stiffness matrices are derived.

The elemental stiffness and mass matrices are

$$[k_e] = \frac{EI}{l^3}\begin{bmatrix} \left(\frac{l}{r}\right)^2 & 0 & 0 & -\left(\frac{l}{r}\right)^2 & 0 & 0 \\ 0 & 12 & 6l & 0 & -12 & 6l \\ 0 & 6l & 4l^2 & 0 & -6l & 2l^2 \\ -\left(\frac{l}{r}\right)^2 & 0 & 0 & \left(\frac{l}{r}\right)^2 & 0 & 0 \\ 0 & -12 & -6l & 0 & 12 & -6l \\ 0 & 6l & 2l^2 & 0 & -6l & 4l^2 \end{bmatrix} \qquad [m_e] = \frac{\gamma l}{420}\begin{bmatrix} 140 & 0 & 0 & 70 & 0 & 0 \\ 0 & 156 & 22l & 0 & 54 & -13l \\ 0 & 22l & 4l^2 & 0 & 13l & -3l^2 \\ 70 & 0 & 0 & 140 & 0 & 0 \\ 0 & 54 & 13l & 0 & 156 & -22l \\ 0 & -13l & -3l^2 & 0 & -22l & 4l^2 \end{bmatrix}$$

where the terms E is Young's Modulus, I is the area moment of inertia, l is the elemental beam length in inches, γ is the weight density, and r is the radius of gyration. The elemental matrices are partitioned in the following way:

$$\begin{array}{c} \\ x_1 \\ y_1 \\ \theta_1 \\ x_2 \\ y_2 \\ \theta_2 \end{array} \begin{array}{c} \begin{array}{cccccc} x_1 & y_1 & \theta_1 & x_2 & y_2 & \theta_2 \end{array} \\ \begin{bmatrix} & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \end{bmatrix} \end{array}$$

The damping matrix is usually impossible to determine analytically and is typically determined experimentally. Here damping is applied to the system model by one of three ways. The three methods generally used are:

Type (1): Proportional structural damping of the form:

$$[C] = \alpha[K] + \beta[M] \qquad (1)$$

Type (2): Proportional viscous damping of the form:

$$[C] = \alpha[K] \tag{2}$$

Type (3): Frequency-dependent viscous damping of the form:

$$[C] = \left[C_o e^{-a\Omega}\right] \tag{3}$$

Type (1) damping is used in adding damping to structural elements, and Types (2) and (3) are used in adding damping to vibration isolators which are a combination of springs and dampers; adding proportional damping to just the isolators constitutes non-proportional damping for the whole structure.

The equation of motion of these finite elements can be written in terms of their joint displacements as

$$[m]_e\{\ddot{u}_i\}_e + [c]_e\{\dot{u}_i\}_e + [k]_e\{u_i\}_e = \{f_i\}_e \tag{4}$$

where $\{u_i\}_e$ = axial, lateral, or angular joint displacements

$[m]_e$ = mass matrix of element

$[c]_e$ = damping matrix of element

$[k]_e$ = stiffness matrix of element

$\{f_i\}_e$ = joint forces and moments

Since the elements vary in orientation with respect to the system axis, the elemental mass and stiffness matrices must be transformed into global coordinates. The transformation of the damping matrix is neglected, since the elemental damping matrix is modeled as a function of the transformed elemental mass and stiffness matrix. A method for relating local joint displacements of each element to the global system displacements

must be incorporated. This method is referred to as a coordinate transformation. The elemental mass and stiffness matrices after transformation are in the following form [Ref. 4].

$$[m_e] = \frac{\gamma l}{420}\begin{bmatrix} 140C^2+156S^2 & -16CS & -22lS & 70C^2+54S^2 & 16CS & 13lS \\ -16CS & 140S^2+156C^2 & 22lC & 16CS & 70S^2+54C^2 & -13lC \\ -22lS & 22lC & 4l^2 & -13lS & 13lC & -3l^2 \\ 70C^2+54S^2 & 16CS & -13lS & 140C^2+156S^2 & -16CS & 22lS \\ 16CS & 70S^2+54C^2 & 13lC & -16CS & 140S^2+156C^2 & -22lC \\ 13lS & -13lC & -3l^2 & 22lS & -22lC & 4l^2 \end{bmatrix}$$

$$[k_e] = \frac{EI}{l^3}\begin{bmatrix} \left(\frac{l}{r}\right)^2 C^2+12S^2 & \left(\frac{l}{r}\right)^2 CS-12CS & -6lS & -\left(\frac{l}{r}\right)^2 C^2-12S^2 & -\left(\frac{l}{r}\right)^2 CS+12CS & -6lS \\ \left(\frac{l}{r}\right)^2 CS-12CS & \left(\frac{l}{r}\right)^2 S^2+12C^2 & 6lC & -\left(\frac{l}{r}\right)^2 CS+12CS & -\left(\frac{l}{r}\right)^2 S^2-12C^2 & 6lC \\ -6lS & 6lC & 4l^2 & 6lS & -6lC & 2l^2 \\ -\left(\frac{l}{r}\right)^2 C^2-12S^2 & -\left(\frac{l}{r}\right)^2 CS+12CS & 6lS & \left(\frac{l}{r}\right)^2 C^2+12S^2 & \left(\frac{l}{r}\right)^2 CS-12CS & 6lS \\ -\left(\frac{l}{r}\right)^2 CS+12CS & -\left(\frac{l}{r}\right)^2 S^2-12C^2 & -6lC & \left(\frac{l}{r}\right)^2 CS-12CS & \left(\frac{l}{r}\right)^2 S^2+12C^2 & -6lC \\ -6lS & 6lC & 2l^2 & 6lS & -6lC & 4l^2 \end{bmatrix}$$

Noting that:

l = elemental beam length in inches

E = Young's modules in psi

I = Area Moment of Inertia in in^4

r = radius of gyration in inches

γ = mass density per unit length in $lb \cdot s^2 / in^2$

C = cos α

S = sin α

Once the finite elements are transformed to global coordinates, the elements are assembled to generate the global mass, stiffness, and damping matrices. The equation of motion for the modeled system in global coordinates is

$$[M]\{\ddot{u}\} + [C]\{\dot{u}\} + [K]\{u\} = \{F\} \tag{5}$$

where $\{u\}$ = axial, lateral, or angular global joint displacements

$[M]$ = global mass matrix

$[C]$ = global damping matrix

$[K]$ = global stiffness matrix

$\{F\}$ = global joint forces and moments

The following example will demonstrate how the global mass and stiffness matrices are generated. Consider the structural system modeled with two beam elements and having no boundary conditions shown in Figure 2.

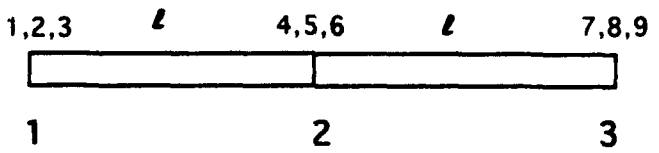


Figure 2. A Beam Modeled Using Two Elements

For this example $\frac{EI}{l^3}$ =1.0 lbs/in, $\frac{\gamma l}{420}$=1.0 $lb \cdot s^2$, l = 1.0 in and r=1.0 in.. Since the beam elements lie horizontally along the x axis, the angle α = 0. The global mass and stiffness matrices are generated from the assembly of the elemental matrices. The elemental matrices are:

$$[k_e] = \begin{bmatrix} 1 & 0 & 0 & -1 & 0 & 0 \\ 0 & 12 & 6 & 0 & -12 & 6 \\ 0 & 6 & 4 & 0 & -6 & 2 \\ -1 & 0 & 0 & 1 & 0 & 0 \\ 0 & -12 & -6 & 0 & 12 & -6 \\ 0 & 6 & 2 & 0 & -6 & 4 \end{bmatrix} \qquad [m_e] = \begin{bmatrix} 140 & 0 & 0 & 70 & 0 & 0 \\ 0 & 156 & 22 & 0 & 54 & -13 \\ 0 & 22 & 4 & 0 & 13 & -3 \\ 70 & 0 & 0 & 140 & 0 & 0 \\ 0 & 54 & 13 & 0 & 156 & -22 \\ 0 & -13 & -3 & 0 & -22 & 4 \end{bmatrix}$$

Referring to Figure 2, the lower bold type numerals represent the node numbering and the upper numbers represent the beam nodal coordinates. Coordinates 1, 2, 3, 4, 5, 6, 7, 8, and 9 are respectively $x_1, y_1, \theta_1, x_2, y_2, \theta_2, x_3, y_3$, and θ_3. Remembering how the matrices are partitioned and noting that coordinates 4, 5 and 6 of beam element 1 are the same coordinates of beam element 2 and thus are shared. The two elemental matrices are assembled together through the shared coordinates. Figure 3 shows the beam element arrangements.

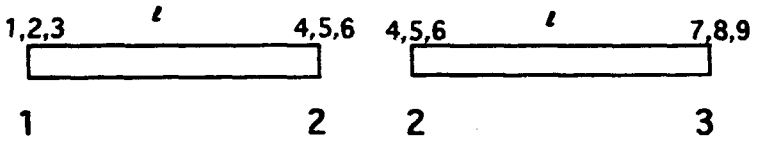


Figure 3. Beam Elements with Node and Global Nodal Coordinate Numbering

For discussion purposes only, the stiffness matrices will be demonstrated, since the mass and damping matrices are generated in the same manner. The elemental stiffness matrices are in the following form.

$$[k_{e1}] = \begin{array}{c} \\ 1 \\ 2 \\ 3 \\ 4 \\ 5 \\ 6 \end{array} \begin{array}{c} \begin{array}{cccccc} 1 & 2 & 3 & 4 & 5 & 6 \end{array} \\ \begin{bmatrix} & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \end{bmatrix} \end{array} \qquad [k_{e2}] = \begin{array}{c} \\ 4 \\ 5 \\ 6 \\ 7 \\ 8 \\ 9 \end{array} \begin{array}{c} \begin{array}{cccccc} 4 & 5 & 6 & 7 & 8 & 9 \end{array} \\ \begin{bmatrix} & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \end{bmatrix} \end{array}$$

The two matrices are combined by adding shared nodal coordinates, this process is determined by the element connectivity. Figure 3 shows that element 1 is coupled to element 2 through global nodal coordinates 4, 5 and 6. These global coordinates are the combination of local coordinates x_2, y_2, θ_2 of element 1 and x_1, y_1, θ_1 of element 2. The resulting matrix is a (9x9) global matrix represented by $[K]$. The size of the global matrix is the number of nodes times the DOF. The global stiffness matrix $[K]$ is shown below with the numbers installed, take special note to the shared coordinates which are additive.

$$[K] = \begin{bmatrix} 1 & 0 & 0 & -1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 12 & 6 & 0 & -12 & 6 & 0 & 0 & 0 \\ 0 & 6 & 4 & 0 & -6 & 2 & 0 & 0 & 0 \\ -1 & 0 & 0 & 2 & 0 & 0 & -1 & 0 & 0 \\ 0 & -12 & -6 & 0 & 24 & 0 & 0 & -12 & 6 \\ 0 & 6 & 2 & 0 & 0 & 8 & 0 & -6 & 2 \\ 0 & 0 & 0 & -1 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & -12 & -6 & 0 & 12 & -6 \\ 0 & 0 & 0 & 0 & 6 & 2 & 0 & -6 & 4 \end{bmatrix}$$

The shared coordinates are demonstrated by looking at the 3x3 partition, rows 4 through 6 and columns 4 through 6. After the global matrix is generated, the boundary conditions are applied. Boundary conditions are determined by coordinate restraints. If a coordinate is restrained then the row and column corresponding to that coordinate are deleted. For example, if in Figure 2, the left end had been fixed, displacements for global coordinates 1, 2, and the slope of coordinate 3 are zero, and therefore the rows and columns corresponding to these coordinates would be deleted resulting in a (6x6) $[K]$ matrix.

The derivation of the equation for a second order linear structural system described in the frequency domain is presented below. The differential equation of motion for a second order linear structural system is written as

$$m\ddot{x} + c\dot{x} + kx = F\sin\Omega t. \tag{6}$$

The solution to equation (6), which is the total system response is

$$X = X_h + X_p \tag{7}$$

where X_h is the real or homogeneous solution and X_p is the particular solution. We consider only steady state harmonic excitation, therefore the particular solution is used as the total solution. The solution is assumed to have the form

$$X = X_p = Xe^{j\Omega t}. \tag{8}$$

Taking the first and second derivatives of X and substituting into equation (6) yields

$$\left(-\Omega^2 mX + j\Omega cX + kX\right)e^{j\Omega t} = Fe^{j\Omega t}. \tag{9}$$

Dividing both sides by $e^{j\Omega t}$ and rearranging equation (9) gives the equation of motion as

$$\left(k - \Omega^2 m + j\Omega c\right)X = F. \tag{10}$$

Writing equation (10) in matrix form gives the equation for second order linear structural systems described in the frequency domain.

$$\left[[K] - \Omega^2[M] + j[C(\Omega)]\right]\{x\} = \{F\} \tag{11}$$

where the vector $\{x\}$ is the set of generalized responses in the global coordinate system. The vector $\{F\}$ contains generalized global forces and moments. $[K]$ and $[M]$ are symmetric, real valued and of order n. The damping matrix $[C(\Omega)]$ is in general, frequency dependent, but here is modeled as a linear proportional combination of the mass and stiffness matrices. Equation (6) is generally written in compact form as

$$[Z(\Omega)]\{x\} = \{F\} \tag{12}$$

where the matrix $[Z(\Omega)]$ is called the system impedance matrix. Equation (12) is the system impedance relationship and represents the dynamic response of the system. The impedance matrix is the dynamic stiffness of the system. The static case is when $\Omega = 0$ and then the system impedance matrix is just the stiffness matrix $[K]$.

The impedance matrix for the assembled beam (Figure 2) is

$$[Z] = [K] - \Omega^2[M]. \tag{13}$$

Equation (13) is reduced because damping $[C(\Omega)]$ is neglected in this example. $[Z(\Omega)]$ is calculated over the frequency band of interest, where Ω is the frequency band of interest in rad/sec.

The frequency response function (FRF) matrix for the assembled beam is

$$[H(\Omega)] = [Z(\Omega)]^{-1}. \tag{14}$$

The FRF matrix allows the calculation of the steady state harmonic response amplitude $\{X\}$ resulting from a harmonic force amplitude $\{F\}$. The frequency response relation is determined by matrix inversion of equation (12), which yields

$$\{X\} = [H(\Omega)]\{F\}. \tag{15}$$

Any element H_{ij} of the frequency response matrix is defined as the dynamic response of motion coordinate i due to a unit harmonic generalized force acting on motion coordinate j. The FRF matrix can be used to represent information about displacements, velocities, accelerations, stress, or strains. For example, if a structure is excited at nodal coordinate 5, then H_{15} is the complex amplitude of the response at nodal coordinate 1 due to a unit harmonic excitation at nodal coordinate 5 at some frequency Ω of interest.

A typical frequency response function plot is shown in Figure 4. The peaks shown in Figure 4 occur at the frequency of peak response. The relationship between the natural undamped frequency, the damped natural frequency, and the frequency of peak response is shown on the following page. The plot shows at what frequencies the structure will have maximum responses and enables the designer to redesign the structure so that the system will have small responses in the frequency bandwidth of interest.

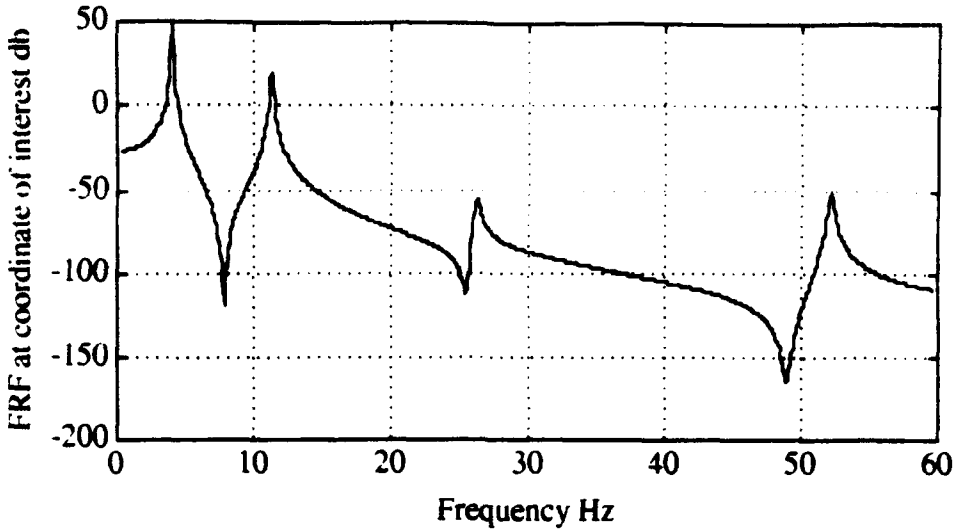


Figure 4. Typical Frequency Response Function Plot

Generally there are three distinct frequencies of interest, the undamped natural frequency, the damped natural frequency, and the frequency of peak response. These frequencies are related by the modal damping factor. The amplitude of the response of a forced vibration can become very large when the frequency of the excitation approaches one of the natural frequencies of the system . This condition where the excitation frequency is the same as one of the natural frequencies is referred to as resonance. When a system vibrates at resonance, the attendant stresses and strains have the potential of causing structural failure. A structural system will have a maximum response when the frequency of excitation is near the undamped natural frequency. If the system has no damping, then the maximum response will occur at the undamped natural frequency. The undamped natural frequency is a function of the system mass and stiffness and is analytically expressed as the solution to the eigensystem

$$[K - \omega^2 M]\{\Phi\} = \{0\} \tag{16}$$

Every real structural system has an infinite number of natural frequencies and mode shapes. The finite modeling of the structural system yields a finite number of eigensolutions or mode shapes and eigenvalues or natural frequencies depending on how many degrees of freedom the structural system is modeled with. Each eigenvector has a corresponding eigenvalue or natural frequency. However, all systems inherently have some degree of damping and the relationship that relates the damped natural frequency to the undamped natural frequency is

$$\omega_d = \omega_{n_i}\sqrt{1-\zeta_i^2} \tag{17}$$

where ζ_i is the modal damping factor for mode i. The damped natural frequency is slightly lower than the undamped natural frequency and a typical damping factor for structural systems is 0.2. The frequency of peak response is the frequency of excitation where the response of the system is maximum. The analytical relation that relates the frequency of peak response to the undamped natural frequency is determined by taking the derivative with respect to $\left(\frac{\omega}{\omega_n}\right)$ of equation (18) and setting it equal to zero.

$$|X| = \frac{Fo/k_i}{\sqrt{\left[1-\left(\frac{\omega}{\omega_{n_i}}\right)^2\right]^2 + \left[2\zeta_i\left(\frac{\omega}{\omega_{n_i}}\right)\right]^2}} \tag{18}$$

Let $\left(\frac{\omega}{\omega_n}\right) = \bar{\omega}$

and substituting back into equation (18) yields

$$\frac{|X|}{Fo/k_i} = \frac{1}{\sqrt{\left[1-\bar{\omega}^2\right]^2 + \left[2\zeta_i\bar{\omega}\right]^2}}. \tag{19}$$

Now performing $\frac{d|X|}{d\bar{\omega}} = 0$

$$-\frac{1}{2}\left[(1-\bar{\omega}^2)^2 + (2\zeta_i\bar{\omega})^2\right]^{-\frac{3}{2}}\left[-4\bar{\omega}(1-\bar{\omega}^2) + 8\zeta_i^2\bar{\omega}\right] = 0 \tag{20}$$

and knowing for equation (20) to equal zero, the numerator must equal zero. Setting the numerator equal to zero

$$2\bar{\omega}(1-\bar{\omega}^2) - 4\zeta_i^2\bar{\omega} = 0 \tag{21}$$

and simplifying

$$\bar{\omega}^2 = 1 - 2\zeta_i^2 \tag{22}$$

and solving for ω , which is the frequency of peak response yields

$$\omega_{pr_i} = \omega_n\sqrt{1-2\zeta_i^2}. \tag{23}$$

It is important when designing a structural system that the excitation frequency is not close to these frequencies or failure of the structural system is likely to occur.

III. FREQUENCY DOMAIN STRUCTURAL SYNTHESIS

The theory presented herein is taken directly and exclusively from references [1, 2, and 3]. The purpose of this thesis is to explore the application of this previously developed theory to the analysis of a submarine cradle structure.

Frequency domain structural synthesis was first presented in 1939 and has evolved to the latest formulation, which was published in *Journal of Sound and Vibration* (1991) [Ref. 1]. The most recent formulation of the theory is a new method for analyzing structural systems [Ref. 3]. This method handles all types of structural models and is more efficient and cost effective compared with traditional finite element solution procedures. Frequency domain structural synthesis refers to substructure coupling and structural modification using frequency response function data. The previously developed formulation for structural synthesis, Ref. [3], is applicable to the static and dynamic structural analysis of direct coupling of substructures, indirect coupling of substructures, modification of substructures, and constraint application. The theory allows the synthesis of displacements, velocities, accelerations, stresses, and strains.

An important feature of the frequency domain formulation is the arbitrary and exact model order reduction possible when performing a synthesis. A finite element method (FEM) when applied to practical problems typically generates between 10^2 to 10^5 degrees-of-freedom (DOF). The frequency domain formulation allows, as a minimum, only those DOF of interest to be included in the analysis. This feature is in fact the reason for the high computational efficiency of the method. Using one of the numerical examples presented in the section "Numerical Examples, " the computing time required for a frequency domain synthesis can be compared with the same analysis using traditional finite element (FE)

procedure. Referring to Example (6) the following count of floating point operations (FLOPS) shows the efficiency of the frequency domain method:

FEM direct assembly: Time - 25876 sec or 431.3 mins

FLOPS - 1.49×10^9

FRF synthesis: Time - 1167 sec or 19.45 mins

FLOPS - 517.2×10^6

This clearly demonstrates that synthesis by FRF is more efficient and better suited for the re-analysis of complex structures with large numbers of DOF. Moreover, the savings in time grows with increasing model size.

There are two major classifications of structural synthesis. These classifications are coupling and modification and each classification can be viewed as direct or indirect. Coupling is defined as the joining of two separate substructures to form one structure and modification is defined as the creation of a new load path in an existing structure. An example of coupling is the coupling of a submarine hull and the machinery support cradle. The hull is modeled as one substructure and the cradle is modeled as another substructure. We want to join these two substructures together to create one complete structural system. This process is known as structural coupling. As an example of the use of structural modification, an analysis of the complete cradle structural system shows that a certain element has unacceptable stresses. By installing an additional support, the stresses become acceptable. This process of changing the structure is known as structural modification. Indirect coupling is the joining of structures with the introduction of an intermediate or interconnecting structural element, referred to as an interconnection impedance element or impedance patch [Ref. 1]. Direct modification can be viewed as the application of a constraint equation to a given structural model; direct coupling is simple substructure synthesis [Ref. 1]. The theory is unique in that it allows any linear structural element to be

used as an interconnection impedance. for example a spring and viscous damper may be installed between two elements of a structure. The synthesis is performed at each frequency of interest, which makes possible the efficient treatment of frequency dependent properties, like the properties in the spring and viscous damper. Frequency domain structural synthesis allows changes to a finite element model without reassembly of the mass, stiffness, or damping matrices.

A. GENERALIZED FREQUENCY RESPONSE

The derivations presented here are taken exclusively from References 1, 2, and 3. The derivations are reproduced with more intermediate steps leading to the final operative equations. We begin the development with the previously derived formulation for a second order linear structural system described in the frequency domain.

The differential equation of motion for a second order linear structural system is written as

$$m\ddot{x} + c\dot{x} + kx = F\sin\Omega t. \tag{6}$$

The solution to equation (6), which is the total system response is

$$X = X_h + X_p \tag{7}$$

where X_h is the real or homogeneous solution and X_p is the particular solution. We consider only steady state harmonic excitation, therefore the particular solution is used as the total solution. The solution is assumed to have the form

$$X = X_p = Xe^{j\Omega t}. \tag{8}$$

Taking the first and second derivatives of X and substituting into equation (6) yields

$$\left(-\Omega^2 mX + j\Omega cX + kX\right)e^{j\Omega t} = Fe^{j\Omega t}. \tag{9}$$

Dividing both sides by $e^{j\Omega t}$ and rearranging equation (9) gives the equation of motion as

$$\left(k - \Omega^2 m + j\Omega c\right)X = F. \tag{10}$$

The dynamic stiffness of the structural system is known as the system impedance which is written as

$$Z(\Omega) = k - \Omega^2 m + j\Omega c. \tag{24}$$

The static system stiffness k is determined by the case where $\Omega = 0$ and the system impedance is

$$Z(\Omega) = k. \tag{25}$$

The matrix notation for the structural system impedance is

$$[Z(\Omega)]\{x\} = \{f\}. \tag{26}$$

The system impedance matrix $[Z(\Omega)]$ is both complex valued and frequency dependent. The general equation for the frequency response structural model is found by taking the matrix inversion of equation (26) and is indicated as

$$\{x(\Omega)\} = [H(\Omega)]\{f(\Omega)\}. \tag{27}$$

$\{x\}$ and $\{f\}$ are vectors of complex valued generalized response and excitation coordinates at a specific frequency Ω, and $[H]$ is the frequency response function (FRF) matrix evaluated at the frequency Ω. In general an element of the FRF matrix is defined by taking the partial derivative of $\{x\}$ with respect to $\{f\}$. Referring to equation (24) and writing the equation for x_1 we get

$$x_1 = H_{11}f_1 + H_{12}f_2 + H_{13}f_3 + \cdots + H_{1n}f_n \tag{28}$$

and taking the partial derivative of equation (28), the general form for an element of the FRF matrix is

$$H_{ij} = \frac{\partial x_i}{\partial f_j} \tag{29}$$

and is defined as the partial derivative of the ith generalized response coordinate with respect to the jth generalized excitation coordinate.

There are other types of frequency response which are classified by the type of coordinates involved. For example, strain-force and stress-force frequency response are defined as

$$\{\varepsilon\} = [H^{\varepsilon}]\{f\} \qquad \{\sigma\} = [H^{\sigma}]\{f\}. \tag{30,31}$$

The difference between equation (27) and the two equations (30,31) is the FRF matrix $[H]$ contains displacement-force information in equation (27) and strain/stress-force information

in equations (30.31). The general element of the strain and stress FRF is determined in the same manner as the displacement-force FRF. The general elements are defined as

$$H_{ij}^{\epsilon} = \frac{\partial \epsilon_i}{\partial f_j} \qquad H_{ij}^{\sigma} = \frac{\partial \sigma_i}{\partial f_j} \tag{32.33}$$

where ϵ_i and σ_i are complex valued strains and stresses at coordinates i at a specific frequency Ω

Here we will show the development of the frequency response function in the modal coordinate system. We start with the differential equation of motion for a second order linear structural system in physical coordinates.

$$m\ddot{x} + c\dot{x} + kx = F(t) \tag{34}$$

where we assume $F(t)$ is of the form $\{\overline{F}\}e^{j\Omega t}$ and the vector $\{\overline{F}\}$ is the set of force amplitudes. Now we apply the linear transformation

$$\{x\} = [\Phi]\{q\} \tag{35}$$

to equation (34) where the vector $\{x\}$ is the set of physical coordinates to be transformed, the vector $\{q\}$ is the set of modal coordinates, and the matrix $[\Phi]$ is the set of mass normalized normal mode shapes. Pre multiplying the transformed equation by $[\Phi]^T$ and using the relation $[\Phi]^T[M][\Phi] = [I]$ yields

$$[I]\{\ddot{q}\} + \begin{bmatrix} \ddots & & \\ & 2\zeta_i\omega_i & \\ & & \ddots \end{bmatrix}\{\dot{q}\} + \begin{bmatrix} \ddots & & \\ & \omega_i^2 & \\ & & \ddots \end{bmatrix}\{q\} = [\Phi]^T\{F\} = \{\mathcal{F}\}. \tag{36}$$

Rewriting equation (36), the differential equation of motion in the modal coordinate system is

$$\ddot{q}_i + 2\zeta_i\omega_i\dot{q} + \omega_i^2 = \mathcal{F}_i(t). \tag{37}$$

Since we assumed steady state harmonic forces, equation (34), the modal forces are also of the form $\{\mathcal{F}\} = \{\bar{\mathcal{F}}\}e^{j\Omega t}$ and the solution is assumed as a steady state harmonic modal response of the form $\{q\} = \{\bar{\mathcal{X}}\}e^{j\Omega t}$. Taking the first and second derivative of $\{q\}$ and substituting back into equat[illegible]37) and simplifying yields

$$\left[\begin{bmatrix} \omega_i^2 & \\ & \ddots \end{bmatrix} - \Omega^2[I] + j\Omega\begin{bmatrix} 2\zeta_i\omega_i & \\ & \ddots \end{bmatrix}\right]\{\bar{\mathcal{X}}\} = \{\bar{\mathcal{F}}\}. \tag{38}$$

Rewriting equation (38)

$$\begin{bmatrix} \omega_i^2 - \Omega^2 + j\Omega 2\zeta_i\omega_i & \\ & \ddots \end{bmatrix}\{\bar{\mathcal{X}}\} = \{\bar{\mathcal{F}}\} \tag{39}$$

and solving equation (39) in terms of the modal response yields

$$\{\bar{\mathcal{X}}\} = \begin{bmatrix} \dfrac{1}{\omega_i^2 - \Omega^2 + j\Omega 2\zeta_i\omega_i} & \\ & \ddots \end{bmatrix}\{\bar{\mathcal{F}}\}. \tag{40}$$

To transform equation (40) back to the physical coordinate system, we use the transformation of modal force, $[\Phi]^T\{F\} = \{\overline{F}\}$, and the transformation of modal coordinates, $\{x\} = [\Phi]\{\overline{x}\}$, to substitute back into equation (40) and simplify. The resulting equation in physical coordinates is

$$\{x\} = [\Phi]\begin{bmatrix} \ddots & & \\ & \dfrac{1}{\omega_i^2 - \Omega^2 + j\Omega 2\zeta_i\omega_i} & \\ & & \ddots \end{bmatrix}[\Phi]^T\{F\}. \tag{41}$$

Remembering the general form of the frequency response, equation (27), the frequency response function $[H(\Omega)]$ in terms of the system modal information is

$$[H(\Omega)] = [\Phi]\begin{bmatrix} \ddots & & \\ & \dfrac{1}{\omega_i^2 - \Omega^2 + j\Omega 2\zeta_i\omega_i} & \\ & & \ddots \end{bmatrix}[\Phi]^T \tag{42}$$

and any specific element of H is given by

$$H_{ij} = \sum_{r=1}^{n\ \text{modes}} \frac{\phi_i^r\phi_j^r}{\omega_r^2 - \Omega^2 + j\Omega 2\zeta_r\omega_r}. \tag{43}$$

B. MATRIX PARTITIONING

First we will define the classification of coordinates. *Figure 5 represents* two substructures A and B that will be joined together by merging coordinates 2 with 3 and 6 with 7. These coordinates are referred to as connection coordinates and are denoted by the subscript "c". By the definition just stated, the connection coordinates for substructure A is 2 and 6, likewise the connection coordinates for substructure B are 3 and 7. Internal

coordinates, denoted by the subscript "i," are all the remaining coordinates not directly involved in the substructure coupling. In Figure 5, the internal coordinates for substructure A are 1 and 5 and the internal coordinates for substructure B are 4 and 8. The set of all the physical coordinates are denoted as coordinate set "e". If one structure is involved, then the coordinate set "e" contains only the connection and internal coordinates for that structure. If two or more substructures are involved, then the coordinate set "e" contains all the coordinates for all the substructures. The mathematical representation is $e = i \cup c$.

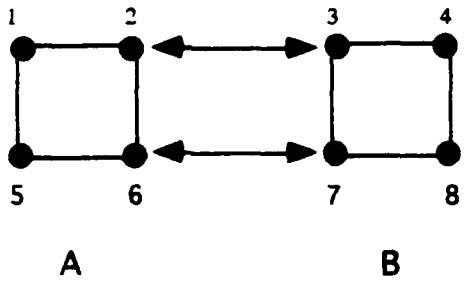


Figure 5. Structural Model with Internal and Connection Coordinates

Referring to the general equation for frequency response, equation (27), and writing it in matrix form with coordinate partitioning as

$$\begin{Bmatrix} x_i \\ x_c \end{Bmatrix} = \begin{bmatrix} H_{ii} & H_{ic} \\ H_{ci} & H_{cc} \end{bmatrix} \begin{Bmatrix} f_i \\ f_c \end{Bmatrix} \tag{44}$$

where x_i and f_i are a set of generalized responses and excitations at the internal coordinates and x_c and f_c are a set of generalized response and excitation at the connection coordinates. One of the special features about the frequency response is that in addition to response information, we can also determine other information at the same time, for example, stresses. We can append a set of stress coordinates and then equation (44) becomes

$$\begin{Bmatrix} \sigma \\ x_i \\ x_c \end{Bmatrix} = \begin{bmatrix} H_{\sigma i} & H_{\sigma c} \\ H_{ii} & H_{ic} \\ H_{ci} & H_{cc} \end{bmatrix} \begin{Bmatrix} f_i \\ f_c \end{Bmatrix}. \tag{45}$$

The stress coordinates will allow the direct calculation of synthesized system stress.

The generalized excitations are partitioned into internal and external excitations. Referring to Figure 5, and looking at the internal coordinates, for example coordinate 1, it is obvious that the only force possible on this coordinate is an externally applied force. Since the internal coordinates do not participate in synthesis, there are no coupling forces present on internal coordinates. The connection coordinates, for example, coordinate 2 in Figure 5, may experience both externally applied forces and coupling forces which are established through synthesis. Therefore

$$f_c = f_c^{ext} + f_c^{cpl} \tag{46}$$

and by definition of the internal coordinates

$$f_i = f_i^{ext} \tag{47}$$

Introducing equations (46 and 47) into equation (45) allows for the expansion of equation (45) as

$$\begin{Bmatrix} \sigma \\ \dot{x}_i \\ \dot{x}_c \\ \ddot{x}_c \end{Bmatrix} = \left[\begin{array}{cc:c} H_{\sigma i} & H_{\sigma c} & H_{\sigma c} \\ \hdashline H_{ii} & H_{ic} & H_{ic} \\ H_{ci} & H_{cc} & H_{cc} \\ \hdashline H_{ci} & H_{cc} & H_{cc} \end{array}\right] \begin{Bmatrix} f_i^{ext} \\ f_c^{ext} \\ f_c^{cpl} \end{Bmatrix} \tag{48}$$

where the asterisk superscript denotes a synthesized quantity due to the fact that we have introduced the forces of synthesis, f_c^{cpl}. Note that with the introduction of equations (46 and 47), a redundant equation, the fourth row of equation (48), has been appended. Using the definition of the set "e", equation (48) is written in the new condensed form

$$\begin{Bmatrix} \sigma \\ x_e \\ x_c \end{Bmatrix}^* = \begin{bmatrix} H_{\sigma e} & H_{\sigma c} \\ H_{ee} & H_{ec} \\ H_{ce} & H_{cc} \end{bmatrix} \begin{Bmatrix} f_e \\ f_c \end{Bmatrix} \tag{49}$$

where $\{f_e\} = \left[f_i^{ext^T} \quad f_c^{ext^T} \right]^T$. The vector f_e is externally applied forces which may exist at all physical coordinates, and the vector f_C is the coupling forces present only at the connection coordinates.

C. STRUCTURAL MODIFICATION AND INDIRECT SUBSTRUCTURE COUPLING

In this section we will develop the governing equation for structural modification and indirect substructure coupling. As previously defined, structural modification is the creation of redundant load paths within a structure, and indirect coupling is the creation of new load path with a structural element between uncoupled substructures. Indirect coupling and structural modification are confined to connection coordinates. There is only one restriction enforced for these processes. The structural change used for modification or interconnection impedance used for indirect coupling must be described by the following equation

$$\{f_c\} = -\left[K(\Omega) - \Omega^2 M(\Omega) + jC(\Omega)\right]\{x_c^*\} \tag{50}$$

or

$$\{f_c\} = -[Z]\{x_c^*\} \tag{51}$$

where the negative sign shows that the reaction is on the structure to be modified or substructures to be coupled. Equation (50) defines the transformation of forces which is used to transform equation (49). The transformation which operates on equation (49) is

$$\begin{Bmatrix} f_e \\ f_c \end{Bmatrix} = \begin{bmatrix} I & 0 \\ 0 & -Z \end{bmatrix} \begin{Bmatrix} f_e \\ x_c^* \end{Bmatrix}. \tag{52}$$

Substituting equation (52) into equation (49) yields

$$\begin{Bmatrix} \sigma \\ x_e \\ x_c \end{Bmatrix}^* = \begin{bmatrix} H_{\sigma e} & H_{\sigma c} \\ H_{ee} & H_{ec} \\ H_{ce} & H_{cc} \end{bmatrix} \begin{bmatrix} I & 0 \\ 0 & -Z \end{bmatrix} \begin{Bmatrix} f_e \\ x_c^* \end{Bmatrix} \tag{53}$$

then performing matrix multiplication and simplifying equation (53), the resulting form is given by the relationship

$$\begin{Bmatrix} \sigma \\ x_e \\ x_c \end{Bmatrix}^* = \begin{bmatrix} H_{\sigma e} & -H_{\sigma c}Z \\ H_{ee} & -H_{ec}Z \\ H_{ce} & -H_{cc}Z \end{bmatrix} \begin{Bmatrix} f_e \\ x_c^* \end{Bmatrix}. \tag{54}$$

Extracting the third row of equation (54)

$$\{x_c^*\} = [H_{ce}]\{f_e\} - [H_{cc}][Z]\{x_c^*\} \tag{55}$$

and rearranging equation (55)

$$[I + H_{cc}Z]\{x_c^*\} = [H_{ce}]\{f_e\} \tag{56}$$

and solving equation (56) in terms of $\{x_c^*\}$ yields

$$\{x_c^*\} = [I + H_{cc}Z]^{-1}[H_{ce}]\{f_e\}. \tag{57}$$

Extracting the second row of equation (54)

$$\{x_e^*\} = [H_{ee}]\{f_e\} - [H_{ec}][Z]\{x_c^*\} \tag{58}$$

and substituting equation (57) into equation (58)

$$\{x_e^*\} = [H_{ee}]\{f_e\} - [H_{ec}][Z]\left([I + H_{cc}Z]^{-1}[H_{ce}]\{f_e\}\right) \tag{59}$$

now, using the known relation that

$$\{x_e^*\} = [H_{ee}^*]\{f_e\} \tag{60}$$

and substituting equation (60) into equation (59) gives the following relation

$$[H_{ee}]^*\{f_e\} = [H_{ee}]\{f_e\} - [H_{ec}][Z][I + H_{cc}Z]^{-1}[H_{ce}]\{f_e\} \tag{61}$$

and rearranging equation (61) and setting it equal to zero

$$[H_{ee}]^*\{f_e\} - [H_{ee}]\{f_e\} + [H_{ec}][Z][I + H_{cc}Z]^{-1}[H_{ce}]\{f_e\} = \{0\} \tag{62}$$

and rewriting equation (62) as

$$\left[\left[H_{ee}\right]^{\bullet} - \left[H_{ee}\right] + \left[H_{ec}\right]\left[Z\right]\left[I + H_{cc}Z\right]^{-1}\left[H_{ce}\right]\right]\left\{f_e\right\} = \left\{0\right\}. \tag{63}$$

Since by definition $\left\{f_e\right\} \neq \left\{0\right\}$, then

$$\left[\left[H_{ee}\right]^{\bullet} - \left[H_{ee}\right] + \left[H_{ec}\right]\left[Z\right]\left[I + H_{cc}Z\right]^{-1}\left[H_{ce}\right]\right] = \left[0\right] \tag{64}$$

and solving equation (64) in terms of $\left[H_{ee}\right]^{\bullet}$ yields

$$\left[H_{ee}\right]^{\bullet} = \left[H_{ee}\right] - \left[H_{ec}\right]\left[Z\right]\left[I + H_{cc}Z\right]^{-1}\left[H_{ce}\right]. \tag{65}$$

Now we will simplify the third term of equation (65). Extracting the following portion

$$\left[Z\right]\left[I + H_{cc}Z\right]^{-1}$$

factoring the inverse term

$$\left[Z\right]\left[\left(Z^{-1} + H_{cc}\right)Z\right]^{-1}$$

and applying the identity $(ab)^{-1} = a^{-1}b^{-1}$

$$\left[Z\right]\left[Z^{-1}\left(Z^{-1} + H_{cc}\right)^{-1}\right]$$

and then simplifying yields

$$\left[Z^{-1} + H_{cc}\right]^{-1}.$$

Substituting the above simplified portion back into equation (65) and performing the same process on the first row of equation (54) yields the final operative equation for structural modification and indirect coupling

$$\begin{bmatrix} H_{oe} \\ H_{ee} \end{bmatrix} = \begin{bmatrix} H_{oe} \\ H_{ee} \end{bmatrix} - \begin{bmatrix} H_{oc} \\ H_{ec} \end{bmatrix} \left[Z^{-1} + H_{cc}\right]^{-1} \left[H_{ce}\right] \tag{66}$$

Note for the static case when $\Omega = 0$, the impedance matrix $[Z] = [K]$ and $[K]$ is a singular matrix which is not invertible, therefore equation (66) is not valid and a form of the equation which does not require the matrix inversion of $[Z]$ must be used. The following equation is for the static case when $\Omega = 0$.

$$\begin{bmatrix} H_{oe} \\ H_{ee} \end{bmatrix} = \begin{bmatrix} H_{oe} \\ H_{ee} \end{bmatrix} - \begin{bmatrix} H_{oc} \\ H_{ec} \end{bmatrix} [Z] \left[I + H_{cc} Z\right]^{-1} \left[H_{ce}\right] \tag{67}$$

Terms on the right side of equations (66 and 67) are pre-synthesized values and the left hand side is the synthesized values. The matrix $[Z]$ describes the modification to be made for structural modification and can be negative if the modification to be made is the removal of a structural modification, or it describes the new load path between two structures for indirect coupling. The quantity $\left[H_{oe}\right]$ allows for the direct calculation of stress due to externally applied loads in the synthesized structure. Stress frequency response could be replaced with strain or other structural frequency response.

D. SUBSTRUCTURE COUPLING AND CONSTRAINT IMPOSITION

In this section, the development of the theory of direct substructure coupling using boolean mapping matrices is shown. A formal discussion of the mapping matrix is presented in the next section. The development of this theory also applies to constraint imposition. Substructure coupling involves the joining of two or more separate substructures where constraint imposition involves one structure, the coupled structure. Constraint imposition is the application of two conditions on the synthesized connection coordinates. The first condition being force equilibrium where the summation of forces on a coordinate are equal to zero and the second condition being compatibility where the displacement of the synthesized coordinates are equal to zero. Compatibility is interpreted as the connection coordinate from the first substructure most have the same displacement as the connection coordinate from the second substructure in order for them to be merged as a single coordinate.

We extract the third row from equation (49) which is shown here again for reader convenience.

$$\begin{Bmatrix} \sigma \\ x_e \\ x_c \end{Bmatrix}^{\bullet} = \begin{bmatrix} H_{\sigma e} & H_{\sigma c} \\ H_{ee} & H_{ec} \\ H_{ce} & H_{cc} \end{bmatrix} \begin{Bmatrix} f_e \\ f_c \end{Bmatrix} \tag{49}$$

The third row of equation (49) is

$$\{x_c\}^{\bullet} = [H_{ce}]\{f_e\} + [H_{cc}]\{f_c\}. \tag{68}$$

We construct the conditions for equilibrium and compatibility to be imposed on the connection coordinates. Figure 6 shows two connection coordinates from two

substructures and Figure 7 shows the equilibrium and compatibility conditions applied to the merged connection coordinate of the synthesized structure.

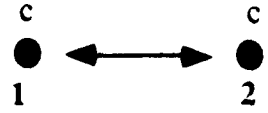


Figure 6. Connection Coordinates from Two Substructures

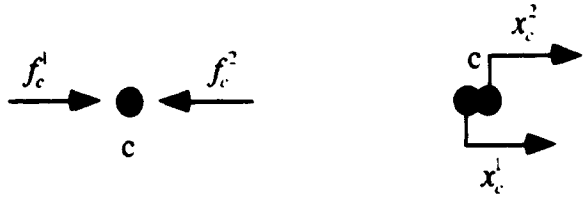


Figure 7. Merged Connection Coordinate

Referring to Figure 7, we write the equilibrium equation for the pair of connection coordinates shown in Figure 6 as

$$f_c^1 + f_c^2 = 0 \tag{69}$$

where the superscript denotes the substructure and the compatibility equation for the merged connection coordinates is

$$x_c^1 - x_c^2 = 0. \tag{70}$$

Converting equations (69 and 70) into the general equations which will encompass all the connection coordinates. The general form uses the mapping matrix to relate each pair of

coordinates. Noting that $f_c^2 = -f_c^1$ and $x_c^1 = x_c^2$, we can write the general equations. The general equation for the force equilibrium is

$$\{f_e\} = [M]\{\tilde{f}_c\} \tag{71}$$

where the vector $\{\tilde{f}_c\}$ represents the arbitrarily selected independent subset of the connection coordinates. Noting that the mapping matrix $[M]$ must remain constant for the constraint imposition to hold, the general form of the compatibility equation is

$$\{\tilde{x}_c\} = [M]^T\{x_c\} = \{0\} \tag{72}$$

where the vector $\{\tilde{x}_c\}$ represents the compatibility for the pairs of connection coordinates. This vector is the zero vector.

The transformation equations that operate on equation (49) are derived from the general equilibrium and compatibility equations and are of the form

$$\begin{Bmatrix} f_e \\ f_c \end{Bmatrix} = \begin{bmatrix} I & 0 \\ 0 & M \end{bmatrix} \begin{Bmatrix} f_e \\ \tilde{f}_c \end{Bmatrix} \tag{73}$$

$$\begin{Bmatrix} \sigma \\ x_e \\ \cdots \\ \tilde{x}_c \end{Bmatrix}^{\bullet} = \left[\begin{array}{c|c} I & 0 \\ \hline 0 & M^T \end{array}\right] \begin{Bmatrix} \sigma \\ x_e \\ \cdots \\ x_c \end{Bmatrix}^{\bullet}. \tag{74}$$

Substituting equation (73 and 74) into equation (49)

$$\begin{Bmatrix} x_e \\ \tilde{x}_c \end{Bmatrix}^{\bullet} = \begin{bmatrix} I & 0 \\ 0 & M^T \end{bmatrix} \begin{bmatrix} H_{ee} & H_{ec} \\ H_{ce} & H_{cc} \end{bmatrix} \begin{bmatrix} I & 0 \\ 0 & M \end{bmatrix} \begin{Bmatrix} f_e \\ \tilde{f}_c \end{Bmatrix} \tag{75}$$

noting that we are using the displacement frequency response only for derivation purposes. and then performing matrix multiplication and simplifying equation (75). The resulting form is given by equation (76).

$$\begin{Bmatrix} x_c \\ \tilde{x}_c \end{Bmatrix} = \begin{bmatrix} H_{cc} & H_{cc}M \\ H_{cc}M^T & M^T H_{cc} M \end{bmatrix} \begin{Bmatrix} f_c \\ \tilde{f}_c \end{Bmatrix}. \tag{76}$$

Extracting the second row of equation (76)

$$\{\tilde{x}_c\} = [H_{cc}M^T]\{f_c\} + [M^T H_{cc} M]\{\tilde{f}_c\} \tag{77}$$

and rewriting the second term of the left hand side

$$[M^T H_{cc} M] = [\tilde{H}_{cc}]$$

and enforcing compatibility between pairs of connection coordinates,

$$\{\tilde{x}_c\} = \{0\} \tag{72}$$

equation (70) becomes

$$\{0\} = [H_{cc}M^T]\{f_c\} + [\tilde{H}_{cc}]\{\tilde{f}_c\}. \tag{78}$$

Rearranging equation (78)

$$[\tilde{H}_{cc}]\{\tilde{f}_c\} = -[H_{cc}M^T]\{f_c\} \tag{79}$$

and pre multiplying both sides of equation (79) by $\left[\tilde{H}_{cc}\right]^{-1}$ yields

$$\left\{\tilde{f}_c\right\} = -\left[H_{ce}M^T\right]\left[\tilde{H}_{cc}\right]^{-1}\left\{f_e\right\}. \tag{80}$$

Extracting the first row of equation (76)

$$\left\{x_e\right\}^* = \left[H_{ee}\right]\left\{f_e\right\} + \left[H_{ec}M\right]\left\{\tilde{f}_c\right\} \tag{81}$$

and substituting equation (80) into equation (81)

$$\left\{x_e\right\}^* = \left[H_{ee}\right]\left\{f_e\right\} - \left[H_{ec}M\right]\left[H_{ce}M^T\right]\left[\tilde{H}_{cc}\right]^{-1}\left\{f_e\right\} \tag{82}$$

and substituting the known relation

$$\left\{x_e^*\right\} = \left[H_{ee}^*\right]\left\{f_e\right\} \tag{60}$$

for the right hand side of equation (82)

$$\left[H_{ee}\right]^*\left\{f_e\right\} = \left[H_{ee}\right]\left\{f_e\right\} - \left[H_{ec}M\right]\left[H_{ce}M^T\right]\left[\tilde{H}_{cc}\right]^{-1}\left\{f_e\right\} \tag{83}$$

and dividing both sides by $\left\{f_e\right\}$ yields the operative equation for direct substructure coupling

$$\left[H_{ee}\right]^* = \left[H_{ee}\right] - \left[H_{ec}\right]\left[M\right]\left[\tilde{H}_{cc}\right]^{-1}\left[M^T\right]\left[H_{ce}\right] \tag{84}$$

where the terms on the right hand side are frequency response values calculated from the uncoupled structures and the left hand side is the frequency response values for the coupled system.

Performing the same derivation presented above on the first row of equation (49), yields the operative equation for direct substructure coupling with coupled system stress response.

$$\begin{bmatrix} H_{\sigma c} \\ H_{ce} \end{bmatrix}^{T} = \begin{bmatrix} H_{\sigma e} \\ H_{ee} \end{bmatrix} - \begin{bmatrix} H_{\sigma c} \\ H_{ec} \end{bmatrix} [M] \left[\tilde{H}_{cc} \right]^{-1} [M^{T}] [H_{ce}] \tag{85}$$

E. DIRECTED GRAPHS AND MAPPING MATRICES

The theory of direct substructure coupling requires mapping matrices to invoke the constraints of equilibrium and compatibility. The theory developed in the preceding section demonstrated how the mapping matrix represented the conditions of equilibrium and compatibility on the synthesized connection coordinates. The mapping matrix can be constructed from a graph which represents the connectivity that is established when substructures are coupled through synthesis. The general formulation of the mapping matrices using directed graphs presented below is taken directly from reference [3].

The use of equation (85) to perform substructure coupling requires the construction of the mapping matrices, [M]. As was developed in the preceding section, each column of [M] represents a statement of the equilibrium and compatibility which is enforced for each pair of connection coordinates being coupled. We will now monstrate that [M] can be constructed from a graph which is drawn to represent the connectivity to be established through the synthesis.

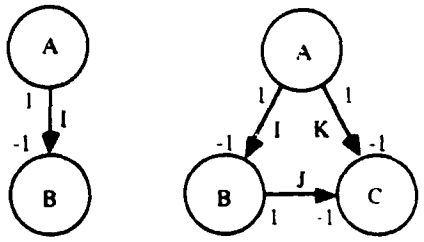


Figure 8. Substructure Couplings and Directed Graphs

Consider the coupling depicted on the left in Figure 8. Substructure "A" is being coupled to substructure "B," through, say, a single pair of connection coordinates, x and x. The coupling of this pair of coordinates creates load path "I." To construct the mapping matrix for this connection "I," we arbitrarily assign a value of "1" to the connection coordinate of substructure "A" and a value "-1" to the connection coordinate of substructure "B." The mapping matrix for this connection is

$$[M] = \begin{matrix} \\ \text{"A"} \\ \text{"B"} \end{matrix} \begin{matrix} \text{"I"} \\ \begin{bmatrix} 1 \\ -1 \end{bmatrix} \end{matrix}$$

Considering now the more complicated coupling on the right of Figure 8, and also acknowledging that in general two substructures are coupled using more than one pair of connection coordinates, we may construct the mapping matrix. Here, the connections "I", "J", and "K" consist of more than one pair of connection coordinates each; these are, in general, sets of connection coordinate pairs. The mapping matrix is

$$[M] = \begin{matrix} \\ \text{"A"} \\ \text{"B"} \\ \text{"C"} \end{matrix} \begin{matrix} \text{"I"} \quad \text{"J"} \quad \text{"K"} \\ \begin{bmatrix} I & 0 & I \\ -I & I & 0 \\ 0 & -I & -I \end{bmatrix} \end{matrix}$$

where each column contains plus/minus identity matrices whose elements correspond to the coupling to be established between each pair of connection coordinates. For example, in column 2 of the above mapping matrix, all connection coordinates associated with substructure "A" are assigned a "1" (i.e. [I]) and they are to be coupled to their counterparts in substructure "C" which have been assigned a "-1" (i.e. –[I]). The coupling of these coordinates constitutes the set of load paths denoted as "J".

The directed graphs and their boolean mapping matrices provide a means of organizing complex couplings, and also provide a framework for the computational implementation of the synthesis, i.e. equation (85). Of course, care must be exercised to insure that all matrices in equation (85) are appropriately partitioned.

An example of using directed graphs to generate the mapping matrix is presented here. Figure 9 shows substructure coupling and the associated directed graph for the load paths created when connection coordinates are coupled through synthesis

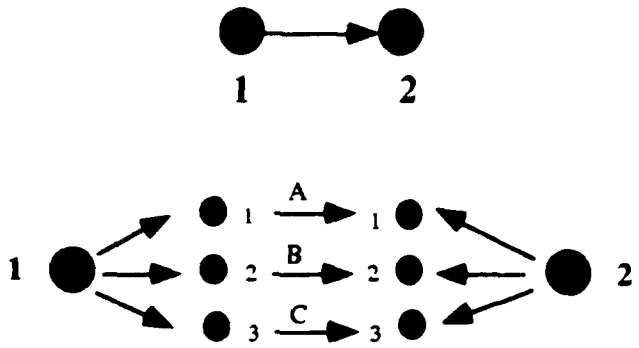


Figure 9. Substructure Coupling Using Directed Graphs

The upper portion of Figure 9 shows two nodes from two substructures that are to be coupled and the lower portion of Figure 9 shows that each node has three degrees of freedom which correlates to three coordinates. Each coordinate of node 1 is synthesized to its corresponding coordinate of node 2. The synthesis of the these coordinates creates load paths "A", "B", and "C". Invoking the constraints of equilibrium and compatibility, the mapping matrix is constructed. Using the equilibrium equation presented earlier

$$\{f_c\} = [M]\{\tilde{f}_c\} \tag{71}$$

where the vector $\{f_c\}$ is the complete set of connection coordinates from both substructures and the vector $\{\tilde{f}_c\}$ is the arbitrary selected subset of connection coordinates to be retained pertaining to the selected substructure. From the equilibrium equation, we get the relationship between the two subsets of connection coordinates, $f_c^2 = -f_c^1$. If we arbitrarily select the connection coordinates from substructure 1 as our set to retain, then we assign a

1 to those coordinates and from the relationship shown above, we assign a -1 to the connection coordinates of substructure 2. The mapping matrix for the system in Figure 9 is determined using equation (71)

$$\begin{matrix} & \text{"A"} & \text{"B"} & \text{"C"} \end{matrix}$$

$$\begin{Bmatrix} 1 \\ 2 \\ 3 \\ \hdashline 1 \\ 2 \\ 3 \end{Bmatrix} = \left[\begin{array}{ccc} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \\ \hdashline -1 & 0 & 0 \\ 0 & -1 & 0 \\ 0 & 0 & -1 \end{array}\right] \begin{Bmatrix} 1 \\ 2 \\ 3 \end{Bmatrix} \tag{86}$$

where

$$[M] = \left[\begin{array}{ccc} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \\ \hdashline -1 & 0 & 0 \\ 0 & -1 & 0 \\ 0 & 0 & -1 \end{array}\right].$$

The upper partition of equation (86) corresponds to arbitrarily selected coordinates of substructure 1 and the lower partition corresponds to connection coordinates of substructure 2. The mapping matrix relates how theses coordinates are connected.

F. MODIFICATION AND INDIRECT COUPLING USING MAPPING MATRICES

This section will show the development of the operative equation of synthesis for indirect substructure coupling and structural modification. There are two classes of synthesis for which mapping matrices are used. The first class is direct substructure coupling which was discussed in Section C and the second class is for indirect substructure

coupling and structural modification. This class of synthesis again uses interconnection impedance to synthesize two substructures or modify an existing structure. The mapping matrix contains the connectivity information corresponding to the equilibrium of the interconnection impedance and the equilibrium of the modification. The interconnection impedance for this method of synthesis has the requirement that the structural element used as an interconnection impedance must be described without mass terms. The interconnection impedance is a function of stiffness and damping. This method is well suited for the synthesis of visco-elastic isolators between substructures.

Visco-elastic isolators are modeled as a combination of a spring and dash pot damper. The isolators are treated as having proportional viscous dampers or as having frequency dependent viscous dampers. A special note here is that adding proportional damping to just the isolators constitutes non-proportional damping for the complete synthesized structure.

We begin the derivation with the description of the structural system, equation (49).

$$\begin{Bmatrix} \sigma \\ x_e \\ x_c \end{Bmatrix}^{\cdot} = \begin{bmatrix} H_{\sigma e} & H_{\sigma c} \\ H_{ee} & H_{ec} \\ H_{ce} & H_{cc} \end{bmatrix} \begin{Bmatrix} f_e \\ f_c \end{Bmatrix} \tag{49}$$

The transformation matrices which operate on equation (49) and lead to the operative equation for indirect coupling and modification using mapping matrices are

$$\begin{Bmatrix} f_e \\ f_c \end{Bmatrix} = \begin{bmatrix} I & 0 \\ 0 & -M\tilde{Z} \end{bmatrix} \begin{Bmatrix} f_e \\ \tilde{x}_c \end{Bmatrix} \tag{87}$$

and

$$\begin{Bmatrix} \sigma \\ x_e \\ \tilde{x}_c \end{Bmatrix}^{\cdot} = \begin{bmatrix} I & 0 & 0 \\ 0 & I & 0 \\ 0 & 0 & M^T \end{bmatrix} \begin{Bmatrix} \sigma \\ x_e \\ x_c \end{Bmatrix}^{\cdot} . \tag{88}$$

The impedance introduced in equation (87) is a reduced system impedance that is massless and is of the form

$$\left[\tilde{Z}\right] = [M]^{+}[Z]\left([M]^{T}\right)^{+}. \tag{89}$$

Using the displacement frequency response of equation (49) and substituting the transformation equations, equation (87 and 88) into equation (49)

$$\begin{Bmatrix} x_e \\ \tilde{x}_c \end{Bmatrix}^{\bullet} = \begin{bmatrix} I & 0 \\ 0 & M^T \end{bmatrix} \begin{bmatrix} H_{ee} & H_{ec} \\ H_{ce} & H_{cc} \end{bmatrix} \begin{bmatrix} I & 0 \\ 0 & -M\tilde{Z} \end{bmatrix} \begin{Bmatrix} f_e \\ \tilde{x}_c \end{Bmatrix} \tag{90}$$

and simplifying equation (90) yields

$$\begin{Bmatrix} x_e \\ \tilde{x}_c \end{Bmatrix}^{\bullet} = \begin{bmatrix} H_{ee} & -H_{ec}M\tilde{Z} \\ M^T H_{ce} & -M^T H_{cc} M\tilde{Z} \end{bmatrix} \begin{Bmatrix} f_e \\ \tilde{x}_c^{\bullet} \end{Bmatrix}. \tag{91}$$

Extracting the second row of equation (91)

$$\{\tilde{x}_c\}^{\bullet} = \left[M^T H_{ce}\right]\{f_e\} - \left[M^T H_{cc} M\tilde{Z}\right]\{\tilde{x}_c^{\bullet}\} \tag{92}$$

and rewriting equation (92) yields

$$\{\tilde{x}_c\}^{\bullet} + \left[M^T H_{cc} M\tilde{Z}\right]\{\tilde{x}_c\}^{\bullet} = \left[M^T H_{ce}\right]\{f_e\}. \tag{93}$$

Equation (93) is rewritten so that the left hand side is a product of sums

$$\left[I + M^T H_{cc} M\tilde{Z}\right]\{\tilde{x}_c\}^* = \left[M^T H_{cc}\right]\{f_c\}. \tag{94}$$

Pre multiplying both sides of equation (94) by $\left[I + M^T H_{cc} M\tilde{Z}\right]^{-1}$ and simplifying yields

$$\{\tilde{x}_c\}^* = \left[I + M^T H_{cc} M\tilde{Z}\right]^{-1}\left[M^T H_{ce}\right]\{f_e\}. \tag{95}$$

Extracting the first row of equation (91)

$$\{x_e\}^* = [H_{ee}]\{f_e\} - [H_{ec}][M]\left[\tilde{Z}\right]\{\tilde{x}_c^*\} \tag{96}$$

and substituting equation (95) into equation (96)

$$\{x_e\}^* = [H_{ee}]\{f_e\} + [H_{ec}][M]\left[\tilde{Z}\right]\left[I + M^T H_{cc} M\tilde{Z}\right]^{-1}\left[M^T H_{ce}\right]\{f_e\} \tag{97}$$

and using the definition of the frequency response

$$\{x_e^*\} = [H_{ee}^*]\{f_e\} \tag{60}$$

to substitute into the left hand side of equation (97) yields

$$[H_{ee}]^* = [H_{ee}] - [H_{ec}][M]\left[\tilde{Z}\right]\left[I + M^T H_{cc} M\tilde{Z}\right]^{-1}\left[M^T\right]H_{ce}]. \tag{98}$$

Noting that

$$\left[M^T H_{cc} M\right] = \left[\tilde{H}_{cc}\right]$$

we will simplify the third term of equation (94). Extracting the following portion

$$\left[\tilde{Z}\right]\left[I + \tilde{H}_{cc}\tilde{Z}\right]^{-1}$$

factoring the inverse term

$$\left[\tilde{Z}\right]\left[(\tilde{Z}^{-1} + \tilde{H}_{cc})\tilde{Z}\right]^{-1}$$

and then simplifying yields

$$\left[\tilde{Z}^{-1} + \tilde{H}_{cc}\right]^{-1}.$$

Substituting the above simplified portion back into equation (98) yields the final operative equation, equation (99), for indirect synthesis and modification using mapping matrices

$$\left[H_{ee}\right]^{*} = \left[H_{ee}\right] - \left[H_{ec}\right]\left[M\right]\left[\tilde{Z}^{-1} + \tilde{H}_{cc}\right]^{-1}\left[M^{T}\right]\left[H_{ce}\right] \tag{99}$$

where the terms on the right hand side are frequency response values calculated from the uncoupled structures and the left hand side is the frequency response values for the coupled system.

Performing the same derivation presented above on the first row of equation (49), yields the operative equation for indirect substructure coupling and modification using mapping matrices with coupled system stress response.

$$\begin{bmatrix} H_{oe} \\ H_{ee} \end{bmatrix}^{*} = \begin{bmatrix} H_{oe} \\ H_{ee} \end{bmatrix} - \begin{bmatrix} H_{oc} \\ H_{ec} \end{bmatrix}\left[M\right]\left[\tilde{Z}^{-1} + \tilde{H}_{cc}\right]^{-1}\left[M^{T}\right]\left[H_{ce}\right] \tag{100}$$

IV. NUMERICAL EXAMPLES

The following numerical examples are provided to give a detailed explanation for each type of synthesis. The results of each example are presented graphically and are compared with the traditional finite element method (FEM) solution.

Three types of damping that are addressed in the numerical examples are:

Type (1): Proportional structural damping of the form:

$$[C] = \alpha[K] + \beta[M] \tag{1}$$

Type (2): Proportional viscous damping of the form:

$$[C] = \alpha[K] \tag{2}$$

Type (3): Frequency-dependent viscous damping of the form:

$$[C] = \left[C_o e^{-a\Omega}\right] \tag{3}$$

Type (1) damping is used in adding damping to a substructure, and Types (2) and (3) are used in adding damping to the isolators which are a combination of spring and dampers. The system impedance matrix Z for these damping types are:

Type (1): $[Z(\Omega)] = [K] - \Omega^2[M] + j[C]$, where $[C] = \alpha[K] + \beta[M]$. (100)

Type (2): $[Z(\Omega)] = [K] - \Omega^2[M] + j\Omega[C]$, where $[C] = \alpha[K]$. (101)

Type (3): $[Z(\Omega)] = [K] - \Omega^2[M] + j\Omega[C(\Omega)]$, where $[C] = \left[C_o e^{-a\Omega}\right]$. (102)

A. EXAMPLE (1): DYNAMIC INDIRECT COUPLING

Consider the structures shown in the following figures. The structure shown in Figure 1.1 will be directly assembled by the finite element method in order to compare a traditional calculation of the frequency response with that synthesized from the substructures shown in Figure 1.2.

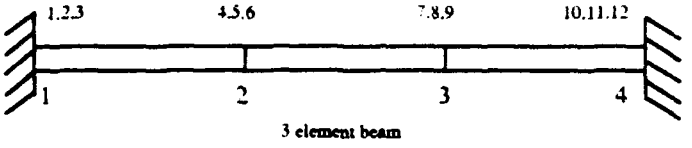


Figure 1.1. Structure Analyzed Using Traditional FE Procedures.

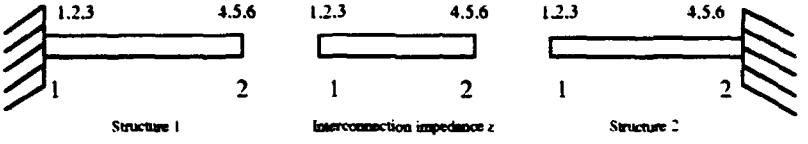


Figure 1.2. Synthesis of Structure

The total structure shown in Figure 1.1 is synthesized from Structure 1 and Structure 2 through the interconnection impedance Z or "new load path." For this example, the following beam parameters will be used:

Young's Modulus $E = 30.0 \times 10^{6}$ psi

Area moment of inertia $I = 0.1666 \times 10^{-3}$ in^4

Cross-sectional area $A = 0.2$ in^2

Weight density $WTD = 0.2832$ lbf/in^3

2 percent proportional structural damping $\alpha = 0.02$

Beam element lengths = 24 in

Proportional structural damping is applied only to structures 1 and 2, and the interconnection impedance Z is undamped. The damping applied in this example was

arbitrarily selected. The synthesis method is not limited to proportional structural damping, any arbitrary linear frequency dependent damping can be used. Referring to Figure 1.2, the system of structures 1 and 2 and the interconnection impedance z are synthesized in the frequency domain to yield exact results as the FEM direct assembly method. The general synthesis equation for dynamic indirect coupling is

$$[H_{ee}]^* = [H_{ee}] - [H_{ec}][Z^{-1} + H_{cc}]^{-1}[H_{ce}] \tag{66}$$

Note again that structures 1 and 2 have proportional structural damping and the interconnection impedance Z is undamped. The general procedure for performing the synthesis is as follows.

The mass and stiffness matrices $[K]$ and $[M]$ for the three substructures (including the middle beam analytically treated as an interconnection impedance) are generated using traditional FEM. Since each structure is comprised of only one beam element, the elemental matrices with boundary conditions applied are the substructure global matrices. The impedance matrix is calculated for each structure as

$$\begin{aligned} [Z_1] &= [K_1] - \Omega^2[M_1] \\ [Z_2] &= [K_2] - \Omega^2[M_2] \\ [Z_z] &= [K_z] - \Omega^2[M_z] \end{aligned} \quad . \tag{103}$$

Note that $[K_1]$ and $[K_2]$ are complex-valued and $[K_z]$ is real-valued. The FRF matrix $[H]$ for structures 1 and 2 is calculated by inverting the impedance matrix $[Z]$. We now have $[H_1]$, $[H_2]$ and $[Z_z]$. Referring to the general synthesis equation provided above, the matrices $[H_{ee}]$, $[H_{ec}]$, $[H_{cc}]$, and $[H_{ce}]$ are generated by assembling $[H_1]$ and $[H_2]$ by appropriate partitioning. $[H_{ee}]$ is the combination of $[H_1]$ and $[H_2]$ and is partitioned by

internal and connection coordinates. Referring to Figure 1.2, after the boundary conditions are applied, coordinates 4, 5, and 6 are renumbered 1, 2, and 3 respectively for structure 1. Structure 2 is unaffected since the boundary conditions remove coordinates 4, 5, and 6 and coordinates 1, 2, and 3 remain the same. The impedance z is unchanged. $[H_{cc}]$ is partitioned in the following manner

$$[H_{cc}] = \begin{bmatrix} H(i,i) & H(i,c) \\ H(c,i) & H(c,c) \end{bmatrix}$$

where the subscript "i" represents the set of internal coordinates and "c" represents the set of connection coordinates. A more detailed representation is

$$[H_{cc}] = \begin{array}{c} \\ i_1 \\ i_2 \\ c_1 \\ c_2 \end{array} \begin{array}{c} \begin{array}{cccc} i_1 & i_2 & c_1 & c_2 \end{array} \\ \left[\begin{array}{cc|cc} H_1(i_1,i_1) & 0 & H_1(i_1,c_1) & 0 \\ 0 & H_2(i_2,i_2) & 0 & H_2(i_2,c_2) \\ \hline H_1(c_1,i_1) & 0 & H_1(c_1,c_1) & 0 \\ 0 & H_2(c_2,i_2) & 0 & H_2(c_2,c_2) \end{array}\right] \end{array}$$

In this representation "i1" denotes the internal coordinates of structure 1 and "c1" denotes the connection coordinates of structure 1. The same principle follows for "i2" and "c2" relating to structure 2. The partitioning for Hec, Hcc, and Hce are

$$[H_{ec}] = \begin{array}{c} \\ i_1 \\ i_2 \\ c_1 \\ c_2 \end{array} \begin{array}{c} \begin{array}{cc} c_1 & c_2 \end{array} \\ \left[\begin{array}{cc} H_1(i_1,c_1) & 0 \\ 0 & H_2(i_2,c_2) \\ \hline H_1(c_1,c_1) & 0 \\ 0 & H_2(c_2,c_2) \end{array}\right] \end{array} \qquad [H_{cc}] = \begin{array}{c} \\ c_1 \\ c_2 \end{array} \begin{array}{c} \begin{array}{cc} c_1 & c_2 \end{array} \\ \left[\begin{array}{cc} H_1(c_1,c_1) & 0 \\ 0 & H_2(c_2,c_2) \end{array}\right] \end{array}$$

$$[H_{ee}] = \begin{matrix} c_1 \\ c_2 \end{matrix} \begin{matrix} \begin{bmatrix} H_1(c_1,i_1) & 0 \\ 0 & H_2(c_2,i_2) \end{bmatrix} & \begin{matrix} i_1 & i_2 \end{matrix} \\ \begin{bmatrix} H_1(c_1,c_1) & 0 \\ 0 & H_2(c_2,c_2) \end{bmatrix} & \begin{matrix} c_1 & c_2 \end{matrix} \end{matrix}$$

In this example there are no internal coordinates. The connection coordinates for structure 1 are (1, 2, 3) and for structure 2 are (1, 2, 3). With the appropriate partitioning complete, the synthesis can now be performed. Using the indirect coupling relation

$$[H_{ee}]^* = [H_{ee}] - [H_{ec}][Z^{-1} + H_{cc}]^{-1}[H_{ce}], \tag{66}$$

structure 1 is synthesized to structure 2 through the "new load path" Z. $[H_{ee}]^*$ is the synthesized FRF relation representing the exact dynamics of the total structure. Finally the FRF relation is calculated over the frequency range 0.1 - 65 Hz and plotted in the figures which follow.

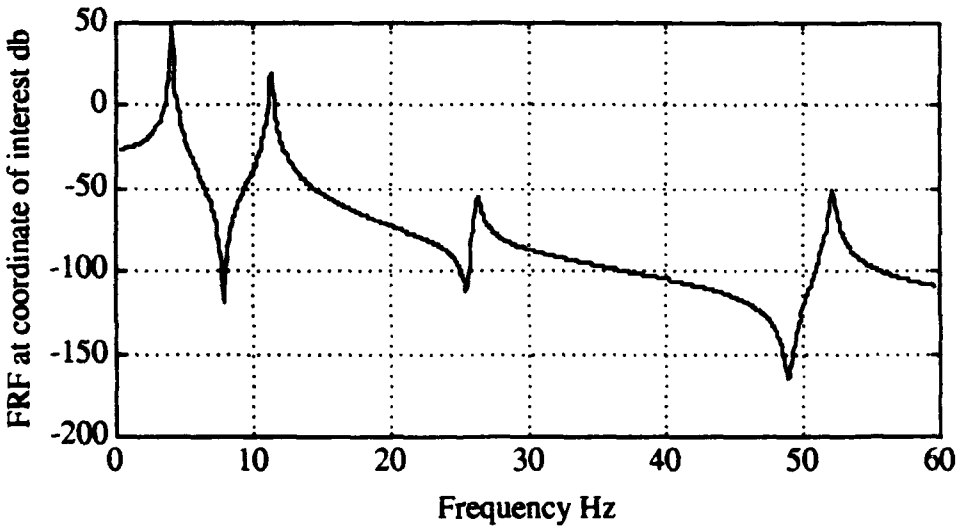


Figure 1.3. Plot of H_{ee} (2,2) from Synthesis

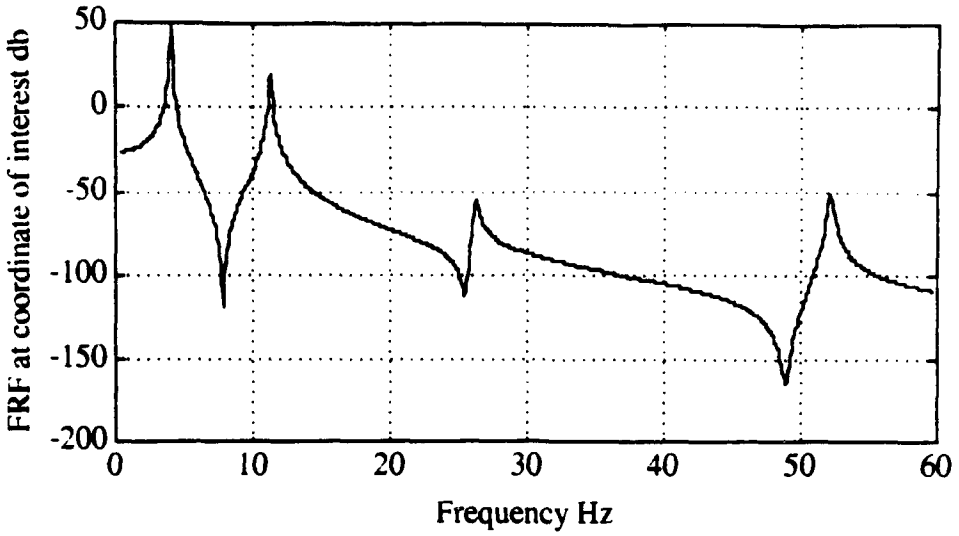


Figure 1.4. Plot H (2,2) from Traditional FE Calculation

Figures 1.3 is a plot of the synthesized $[H_{ee}]^{*}$ matrix, element (2,2), and Figure 1.4 is the same FRF element calculated using the traditional FE procedure. The FRF element plotted in both figures corresponds to coordinate 5 of Figure 1.1, a lateral motion coordinate. Notice both plots are identical, demonstrating that the synthesis procedure provides an exact solution for the synthesized system dynamics. The figures show the first four damped natural frequencies. The FRF plots show the magnitude of the response at coordinate 5 due to a unit excitation of varying frequency at coordinate 5.

B. EXAMPLE (2): DYNAMIC DIRECT COUPLING

Consider the following figures. The structure shown in Figure 2.1 will be directly assembled using FEM for the purpose of comparing with the results obtained by synthesizing structures 1 and 2 of Figure 2.2.

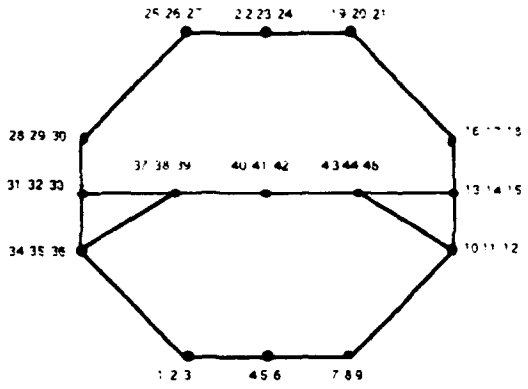


Figure 2.1. Hull-Cradle Structure Analyzed by Traditional FE Techniques

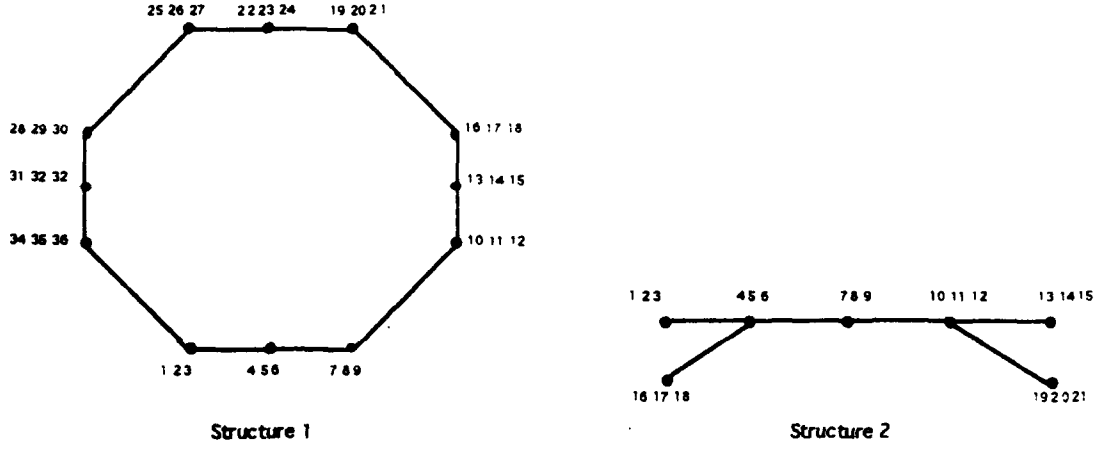


Figure 2.2. Synthesis is Used to Directly Assemble Substructures

Structure 2 will be coupled to structure 1 at coordinates 10, 11, 12, 13, 14, 15, 31, 32, 33, 34, 35, and 36. These coordinates are the connection coordinates and the remaining coordinates are internal coordinates. Coordinates 1, 2, 3, 13, 14, 15, 16, 17, 18 19, 20, and 21 of structure 2 are connection coordinates and the remaining are internal. The following beam element data will be used:

Young's Modulus E = 30.0×10^6 psi

Area moment of inertia I = 0.02083 in^4

Cross-sectional area A = 1 in

Weight density WTD = 0.2832 lbf/in^3

Proportional structural damping (1%) α = 0.01

The proportional structural damping was arbitrarily selected and is applied to both structures. The general equation for dynamic direct coupling is

$$[H_{ee}]^* = [H_{ee}] - [H_{ec}][M]\left[\tilde{H}_{cc}\right]^{-1}[M]^T[H_{ce}]. \tag{84}$$

In this equation, M is the boolean mapping matrix which is used to establish the connectivity between the two substructures for synthesis. The mapping matrix is determined by the connectivity i.e. what is connected to what and by imposing the equilibrium and compatibility relations associated with each pair of coordinates. We can define the mapping matrix by $\{f_c\} = [M]\left\{\tilde{f}_c\right\}$. Where $\{f_c\}$ is a vector of all the connection coordinates of both structures and $\left\{\tilde{f}_c\right\}$ is the arbitrarily selected independent subset of the connection coordinates relating to one of the substructures. We have selected the connection coordinates of structure 1 as the arbitrary subset of connection coordinates. The mapping matrix $[M]$ is a matrix of size (24 x 12) and is depicted as:

$[M]$ =

We will calculate the FRF matrix $[H]$ for both substructures. First the $[K]$ and $[M]$ matrices are generated for each substructure. $[K_1]$ and $[K_2]$ are both complex since proportional structural damping was applied to both structures; again this damping is arbitrary. $[K_1]$ and $[K_2]$ are of the form $[K] = [K + j\alpha K]$. We next form the impedance matrix for each substructure. The impedance matrix is of the form $[Z] = [K] - \Omega^2[M]$. With the impedance matrix generated for each substructure, the FRF matrix H can be calculated by inverting the impedance matrix. This process is done at each frequency of interest. These FRF matrices are required in order to couple the two structures together to form the structure in figure 2.1. Referring to the synthesis equation above, the matrices $[H_{ee}]$, $[H_{ec}]$, $[H_{cc}]$, and $[H_{ce}]$ are formed by combining $[H_1]$ and $[H_2]$ by appropriate partitioning. The partitioning is shown below.

$$[H_{ee}] = \begin{array}{c} \\ i_1 \\ i_2 \\ c_1 \\ c_2 \end{array} \begin{array}{c} \begin{array}{cccc} i_1 & i_2 & c_1 & c_2 \end{array} \\ \left[\begin{array}{cc|cc} H_1(i_1,i_1) & [0] & H_1(i_1,c_1) & [0] \\ [0] & H_2(i_2,i_2) & [0] & H_2(i_2,c_2) \\ \hline H_1(c_1,i_1) & [0] & H_1(c_1,c_1) & [0] \\ [0] & H_2(c_2,i_2) & [0] & H_2(c_2,c_2) \end{array}\right] \end{array}$$

$$[H_{ec}] = \begin{array}{c} \\ c_1 \\ c_2 \end{array} \begin{array}{c} \begin{array}{cccc} i_1 & i_2 & c_1 & c_2 \end{array} \\ \left[\begin{array}{cc|cc} H_1(c_1,i_1) & [0] & H_1(c_1,c_1) & [0] \\ [0] & H_2(c_2,i_2) & [0] & H_2(c_2,c_2) \end{array}\right] \end{array}$$

$$[H_{ec}] = \begin{array}{c} \\ i_1 \\ i_2 \\ c_1 \\ c_2 \end{array} \begin{array}{c} \begin{array}{cc} c_1 & c_2 \end{array} \\ \left[\begin{array}{cc} H_1(i_1,c_1) & [0] \\ [0] & H_2(i_2,c_2) \\ \hline H_1(c_1,c_1) & [0] \\ [0] & H_2(c_2,c_2) \end{array}\right] \end{array} \qquad [H_{cc}] = \begin{array}{c} \\ c_1 \\ c_2 \end{array} \begin{array}{c} \begin{array}{cc} c_1 & c_2 \end{array} \\ \left[\begin{array}{cc} H_1(c_1,c_1) & [0] \\ [0] & H_2(c_2,c_2) \end{array}\right] \end{array}$$

Referring to Figure 2.2, "i1" denotes the set of internal coordinates of structure 1 which are 1, 2, 3, 4, 5, 6, 7, 8, 9, 16, 17, 18, 19, 20, 21, 22, 23, 24, 25, 26, 27, 28, 29, and 30. "c1" denotes the set of connection coordinates of structure 1 which are 10, 11, 12, 13, 14, 15, 31, 32, 33, 34, 35, and 36. "i2" denotes the set of internal coordinates of structure 2 which are 4, 5, 6, 7, 8, 9, 10, 11, and 12. "c2" denotes the set of connection coordinates of structure 2 which are 1, 2, 3, 13, 14, 15, 16, 17, 18, 19, 20, and 21. With the appropriate partitioning complete, the synthesis of structure 1 to structure 2 can be performed using the direct coupling relation

$$[H_{ee}]^{*} = [H_{ee}] - [H_{ec}][M]\left[\tilde{H}_{cc}\right]^{-1}[M]^{T}[H_{ce}]. \tag{84}$$

$[H_{ee}]^{*}$ is the synthesized FRF relation which is the combination of both structures. The synthesis is done over the frequency range of interest and plotted in Figure 2.3. The frequency range for this example was 0.1 to 10.0 Hz. Figure 2.4 is the solution from traditional FE calculations included for direct comparison. Both plots are identical.

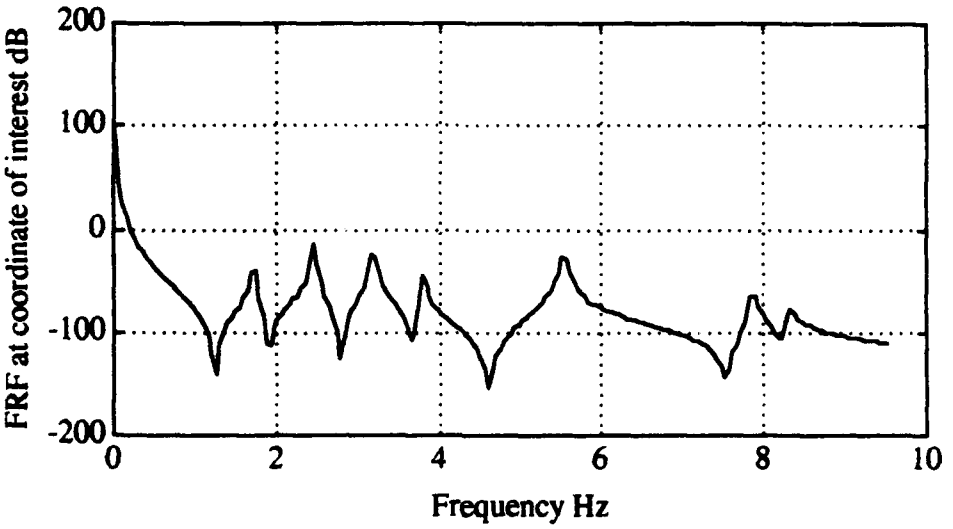


Figure 2.3. Plot of Synthesized $H_{ee}(8,8)$

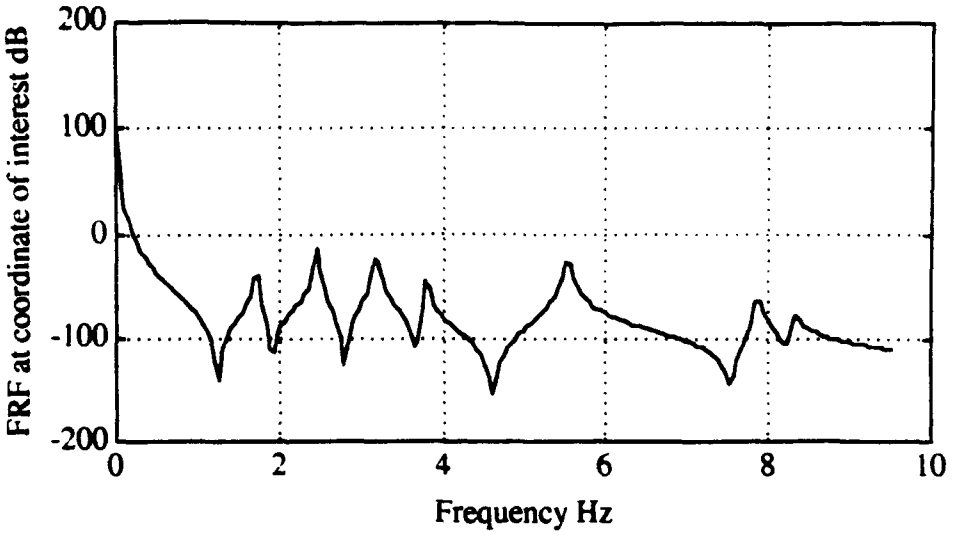


Figure 2.4. Plot of H (8,8) from Traditional FE Calculations.

Figures 2.3 and 2.4 are the plots of the FRF at element (8,8) from the synthesized and FE $[H]$ matrices. This element corresponds to the lateral motion coordinate 8 of Figure 2.1. Notice both plots are identical and both show the first seven damped natural frequencies. The plots show the magnitude of the response of unit amplitude at coordinate 8 due to a unit excitation at varying frequency at coordinate 8. As the frequency of excitation approaches the damped natural frequency, the response approaches infinity.

C. EXAMPLE (3): STRUCTURAL MODIFICATION (REMOVAL OF A BEAM ELEMENT)

Consider the following figures. Figure 3.1 depicts a combined hull-cradle structure which will be directly assembled by traditional FE procedures. Note that the structure in Figure 3.1 has asymmetric reinforcing trusses. The synthesis methodology will be used to arrive at the structural configuration shown on the left of Figure 3.1 by removing the beam

shown on the right of Figure 3.2. The FRF calculated from the FE model (Figure 3.1) will be compared with that calculated using synthesis.

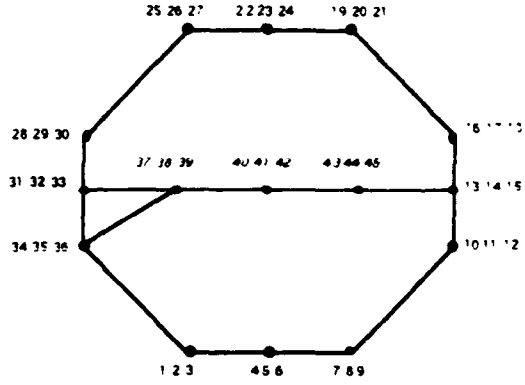


Figure 3.1. Final Hull-Cradle Configuration

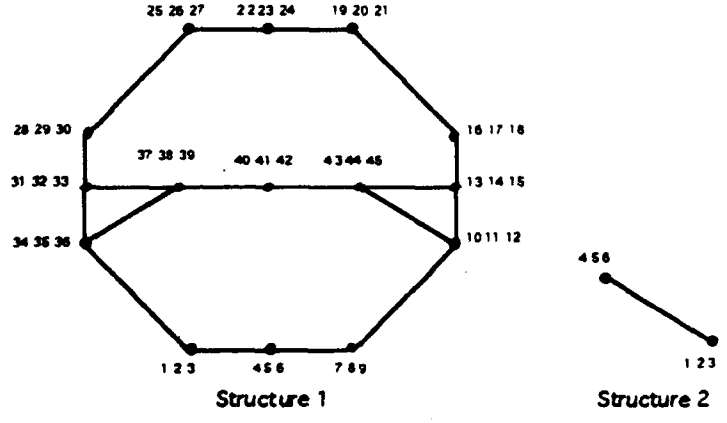


Figure 3.2. Synthesis Used to Remove a Beam Element

Referring to Figure 3.2, structure 1 will be modified by removing the beam, structure 2, located between nodal coordinates 10, 11, 12, 43, 44, and 45. The following beam element data will be used:

Young's Modulus $E = 30.0 \times 10^6$ psi

Area moment of inertia $I = 0.02083$ in^4

Cross-sectional area $A = 1$ in

Weight density WTD = 0.2832 lbf/in^3

1 percent proportional structural damping $\alpha = 0.01$

The proportional structural damping was arbitrarily selected and is applied to both structures. The general equation for dynamic indirect coupling/modification is

$$[H_{ee}]^* = [H_{ee}] - [H_{ec}][H_{cc} - Z^{-1}]^{-1}[H_{ce}]. \tag{66}$$

Note that the sign in the term $[H_{cc} - Z^{-1}]^{-1}$ is opposite from that in the original indirect coupling equation. This is because we are removing the beam element from the structure instead of synthesizing it to the structure. The first step is to generate the $[K]$ and $[M]$ matrices for structure 1 and structure 2. The $[K]$ matrices for both structures are complex since proportional damping was applied. They are of the form $[K] = [K + j\alpha K]$. Next we form the impedance matrices for each structure, $[Z] = [K] - \Omega^2[M]$. This method requires the calculation of the FRF matrix $[H]$ only for the structure to be modified, structure 1 of Figure 3.2. The impedance and the FRF matrices are calculated at the frequency of interest. Once the FRF and impedance matrices are generated, we are ready to partition the FRF matrix. The matrices $[H_{ee}]$, $[H_{ec}]$, $[H_{cc}]$, and $[H_{ce}]$ are formed by partitioning $[H_1]$. The partitioning is shown below.

$$[H_{ee}] = \begin{matrix} \\ i_1 \\ c_1 \end{matrix} \begin{matrix} \begin{matrix} i_1 & c_1 \end{matrix} \\ \left[\begin{array}{c|c} H_1(i_1,i_1) & H_1(i_1,c_1) \\ \hline H_1(c_1,i_1) & H_1(c_1,c_1) \end{array}\right] \end{matrix} \qquad [H_{ce}] = \begin{matrix} \\ c_1 \end{matrix} \begin{matrix} \begin{matrix} i_1 & c_1 \end{matrix} \\ \left[\begin{array}{c|c} H_1(c_1,i_1) & H_1(c_1,c_1) \end{array}\right] \end{matrix}$$

$$[H_{ec}] = \begin{matrix} \\ i_1 \\ c_1 \end{matrix} \begin{matrix} \begin{matrix} c_1 \end{matrix} \\ \left[\begin{array}{c} H_1(i_1,c_1) \\ \hline H_1(c_1,c_1) \end{array}\right] \end{matrix} \qquad [H_{cc}] = \begin{matrix} \\ c_1 \end{matrix} \begin{matrix} \begin{matrix} c_1 \end{matrix} \\ \left[\begin{array}{c} H_1(c_1,c_1) \end{array}\right] \end{matrix}$$

The connection coordinates for structure 1 are 10, 11, 12, 43, 44 and 45. The rest are all treated as internal coordinates. With the appropriate partitioning of $[H_1]$ completed, the removal of the beam from the structure can now be completed by using the correct form of the indirect coupling relation mentioned above. $[H_{ee}]^*$ is the synthesized FRF relation which reflects the removal of structure 2 from of structure 1. This modification is calculated over the frequency range of interest and plotted in Figure 3.3. The frequency range for this example was 0.1 to 7.0 Hz. Figure 3.4 is the solution from the traditional FE procedure and is provided to allow direct comparison of the two solutions. Both plots are identical.

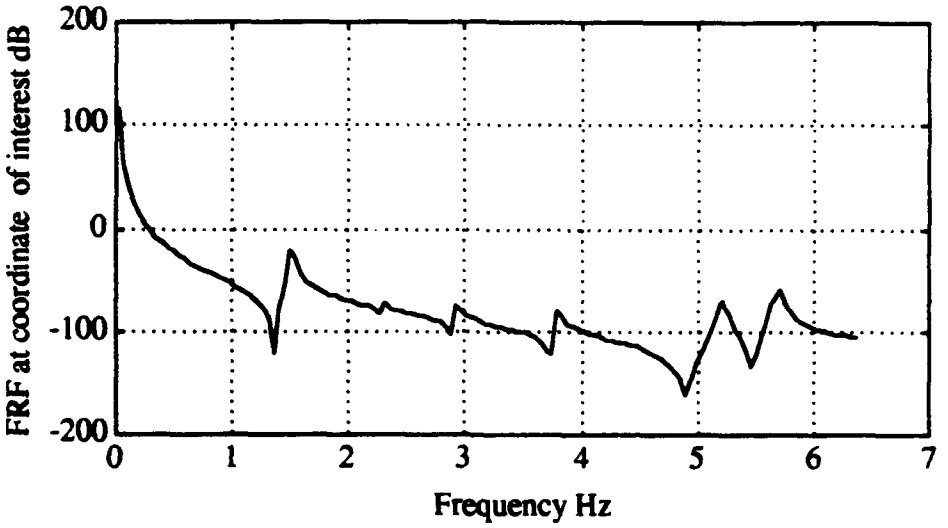


Figure 3.3. Plot of $H_{ee}(11,11)$ as Calculated Using Synthesis

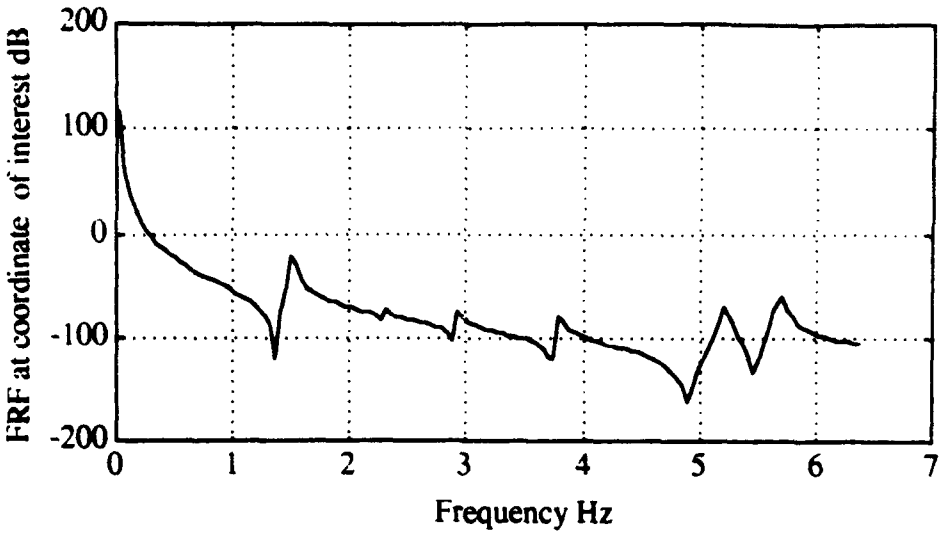


Figure 3.4. Plot of H(14,14) Calculated Using Traditional FE Procedures

Figures 3.3 and 3.4 are the plots of the FRF corresponding to the lateral motion coordinate 14 of Figure 3.1. A special note here is that the element (14,14) of the FRF generated by FEM is the coordinate 14, which corresponds to the element (11,11) of the FRF generated by the indirect coupling relation. The reason for this is because of the partitioning. $[H_{rr}]^*$ is partitioned with internal coordinates first followed by the connection coordinates. Care is required here to ensure the coordinate of interest is actually being used. Notice both plots are identical and show the first six damped natural frequencies. The plots show the magnitude of the response at coordinate 14 due to a unit excitation at varying frequency at coordinate 14. As the frequency of excitation approaches the damped natural frequency, the response approaches infinity.

D. EXAMPLE (4): STRUCTURAL MODIFICATION (ADDITION OF A BEAM)

Consider the following figures. The FRF for the structure shown in Figure 4.1 will be calculated by traditional FE procedures to compare with that calculated using the synthesis procedure to add the beam element, as shown in Figure 4.2.

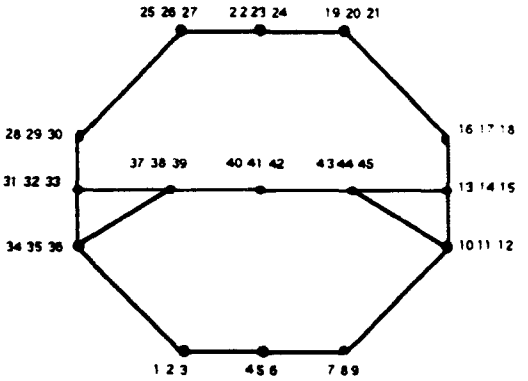


Figure 4.1. Hull-Cradle Structure Analyzed by Traditional FE Techniques.

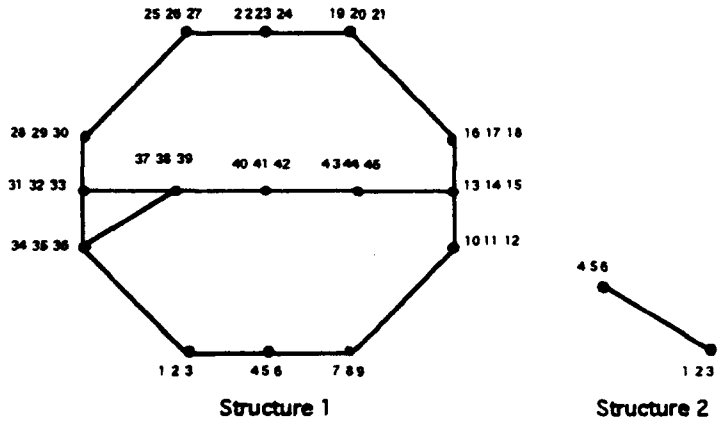


Figure 4.2. Synthesis is Used to Add the Beam Element

Referring to Figure 4.2, structure 1 will be modified by adding the beam, structure 2, at the nodal coordinates 10, 11, 12, 43, 44, and 45. The following beam element data was used:

Young's Modulus $E = 30.0 \times 10^6$ psi

Area moment of inertia $I = 0.02083$ in^4

Cross-sectional area A = 1 in

Weight density WTD = 0.2832 lbf/in^3

Proportional structural damping (1%) α = 0.01

The proportional structural damping was arbitrarily selected and is applied to both structures. The general equation for dynamic indirect coupling/modification is

$$[H_{ee}]^* = [H_{ee}] - [H_{ec}][Z^{-1} + H_{cc}]^{-1}[H_{ce}]. \tag{66}$$

The first step is to generate the $[K]$ and $[M]$ matrices for structure 1 and structure 2. The $[K]$ matrices for both structures are complex since proportional damping was applied. They are of the form $[K] = [K + j\alpha K]$. Next, impedance matrices are formed for each structure as $[Z] = [K] - \Omega^2[M]$. This method requires the calculation of the FRF matrix $[H]$ only for the structure to be modified, structure 1 of Figure 4.2. The impedance and the FRF matrices are calculated at the frequency of interest. Once the FRF and impedance matrices are generated, partition of the FRF matrix is required. The matrices $[H_{ee}]$, $[H_{ec}]$, $[H_{cc}]$, and $[H_{ce}]$ are formed by partitioning $[H_1]$. The partitioning is shown below.

$$[H_{ee}] = \begin{matrix} \\ i_1 \\ c_1 \end{matrix} \begin{matrix} \begin{matrix} i_1 & c_1 \end{matrix} \\ \left[\begin{array}{c|c} H_1(i_1,i_1) & H_1(i_1,c_1) \\ \hline H_1(c_1,i_1) & H_1(c_1,c_1) \end{array}\right] \end{matrix} \qquad [H_{ce}] = \begin{matrix} \\ c_1 \end{matrix} \begin{matrix} \begin{matrix} i_1 & \quad c_1 \end{matrix} \\ \left[H_1(c_1,i_1) \mid H_1(c_1,c_1)\right] \end{matrix}$$

$$[H_{ec}] = \begin{matrix} \\ i_1 \\ c_1 \end{matrix} \begin{matrix} \begin{matrix} c_1 \end{matrix} \\ \left[\begin{array}{c} H_1(i_1,c_1) \\ \hline H_1(c_1,c_1) \end{array}\right] \end{matrix} \qquad [H_{cc}] = \begin{matrix} \\ c_1 \end{matrix} \begin{matrix} \begin{matrix} c_1 \end{matrix} \\ \left[H_1(c_1,c_1)\right] \end{matrix}$$

The connection coordinates for structure 1 are 10, 11, 12, 43, 44, and 45. The rest are all treated as internal coordinates. With the appropriate partitioning of $[H_1]$ completed, the synthesis of the beam to the structure can now be completed by using the correct form of

the indirect coupling relation mentioned above. $[H_{ee}]^*$ is the modified FRF relation which is the combination of structure 1 and the added element, structure 2. The synthesis is performed over the frequency range of interest and plotted in Figure 4.3. The frequency range for this example is 0.1 to 8.5 Hz. Figure 4.4 is the solution from a traditional FE calculation for direct comparison of the two solutions. Both plots are identical.

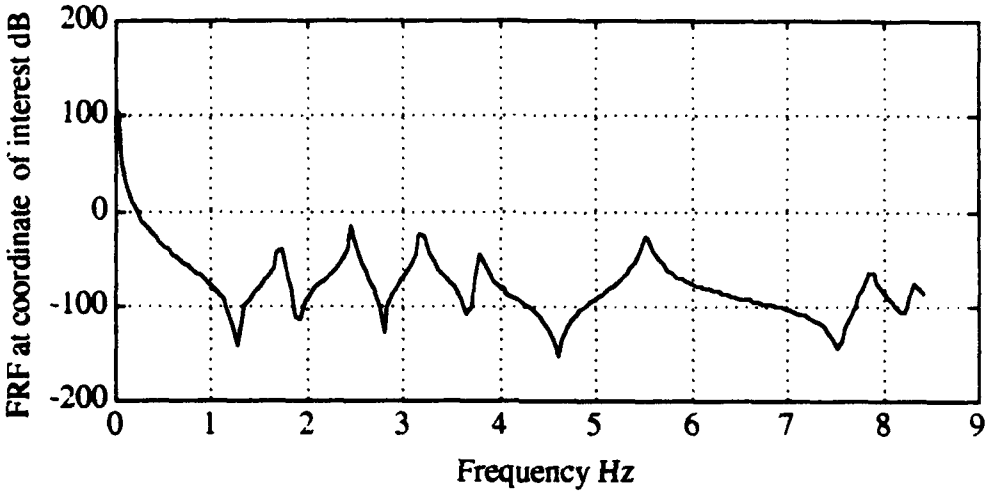


Figure 4.3. Plot of Synthesized FRF Element $H_{ee}(8.8)$

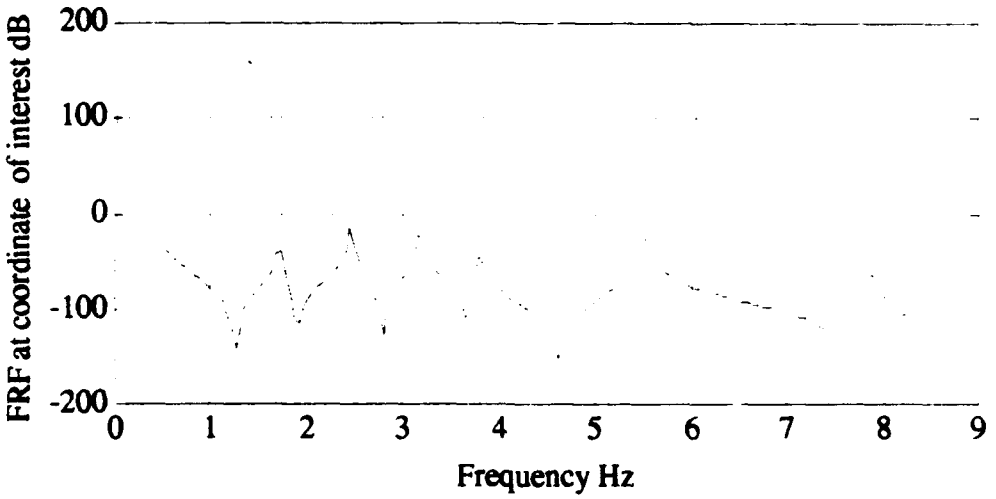


Figure 4.4. Plot of H (8,8) Calculated Using Traditional FE Procedures.

Figures 4.3 and 4.4 are the plots of the FRF corresponding to the lateral motion coordinate 8 of Figure 4.1. A special note here is that the element (8.8) of the FRF generated by FEM corresponds to the coordinate 8, as does the element (8.8) of the FRF generated by the indirect coupling relation. This is different from the previous example. The reason for this is because of the partitioning. $[H_{rr}]^*$ is partitioned with internal coordinates first followed by the connection coordinates. Care is required here to ensure the coordinate of interest is actually being used. Notice both plots are identical and show the first six damped natural frequencies. The plots show the magnitude of the response at coordinate 8 due to a unit excitation at varying frequency at coordinate 8. As the frequency of excitation approaches the damped natural frequency, the response approaches infinity.

E. EXAMPLE (5): INDIRECT COUPLING WITH ISOLATORS

Consider the following figures. The FRF for the structure shown in Figure 5.1 will be calculated using traditional FE procedures to compare with the FRF calculated using the synthesis method. The synthesis will combine the various components shown in Figure 5.2. In this figure, the hull model (structure 1) will be coupled to the cradle model (structure 2). Note that this example demonstrates that the synthesis procedure easily and exactly treats problems with non-proportional damping, a truly unique feature of the methodology.

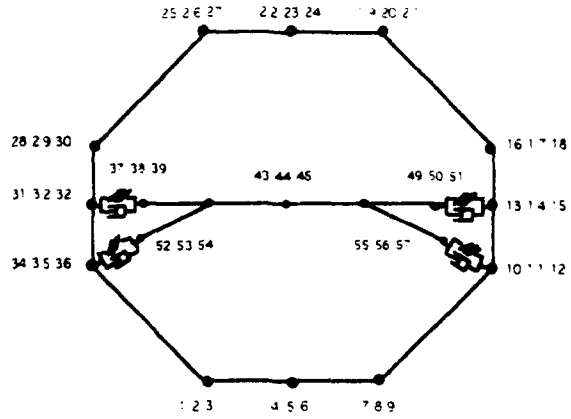


Figure 5.1. Traditional FE Procedures are Used to Calculate FRF for the Combined Hull-Isolator-Cradle Structural System.

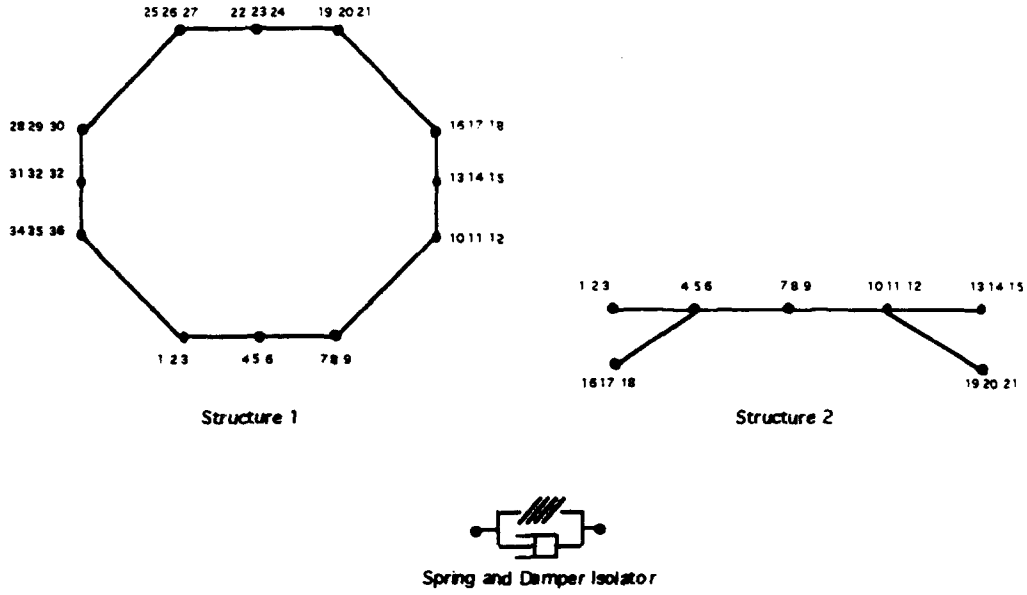


Figure 5.2. Total Hull-Isolator-Cradle System is Synthesized from Components.

Referring to Figure 5.2, structure 1, structure 2, and four spring-damper isolator sets will be synthesized together to form the system in Figure 5.1. Each isolator set consists of three spring-damper isolators, one for each connection coordinate. The connection coordinates for structure 1 are 10, 11, 12, 13, 14, 15, 31, 32, 33, 34, 35, and 36. The remaining

coordinates are internal coordinates. Coordinates 1, 2, 3, 13, 14, 15, 16, 17, 18 19, 20, and 21 of structure 2 are connection coordinates and the remaining are internal. For this structural synthesis method, the spring-damper isolators are treated as a lumped system (with no physical dimensions) installed at the connection coordinates. The connection coordinates do not merge into one but are joined by way of the isolators. The following beam element data will be used:

Young's Modulus E = 30.0×10^6 psi

Area moment of inertia I = 0.02083 in^4

Cross-sectional area A = 1 in

Weight density WTD = 0.2832 lbf/in^3

Proportional structural damping (2%) α = 0.02

Proportional viscous damping (2%) β = 0.02

Isolator spring constant k = 25 lbs/in

The proportional structural damping was arbitrarily selected and is applied to both structure 1 and 2 of Figure 5.2. The proportional viscous damping used for the damper in the isolator is arbitrary and is not limited to being proportional but could be any frequency dependent function. For our example the isolator is of the analytic form $[k + j\Omega\beta k]$ where $j = \sqrt{-1}$. Recalling the impedance relation $[Z(\Omega)] = [K] - \Omega^2[M] + j\Omega[C]$, $[C]$ is the proportional viscous damping, $[\beta k]$. The operative equation for indirect coupling with mapping matrices is

$$[H_{ee}]^* = [H_{ee}] - [H_{ec}][M]\left[\tilde{Z}^{-1} + \tilde{H}_{cc}\right]^{-1}[M]^T[H_{ce}], \tag{99}$$

where $\left[\tilde{Z}\right] = [M]^{\dagger}[Z][M]^{T^{\dagger}}$ and $\left[\tilde{H}_{cc}\right] = [M]^T[H_{cc}][M]$.

Note that $[\tilde{Z}]$ reduces to $[I](k + j\Omega\beta k)$ and its size is (12 x 12). The boolean mapping matrix $[M]$ is determined the same way as explained in example two. The connection coordinates for structure 1 and structure 2 are listed above. We can define the mapping matrix by $\{f_c\} = [M]\{\tilde{f}_c\}$. Where $\{f_c\}$ is a vector of all the connection coordinates of both structures and $\{\tilde{f}_c\}$ is the arbitrarily selected independent subset of the connection coordinates relating to one of the substructures. We have selected structure 1 as the arbitrary subset of connection coordinates. The mapping matrix $[M]$ is a matrix of size (24 x 12) and is

$$[M] = \text{————————}.$$

The FRF matrix $[H]$ for both substructures is required. First the $[K]$ and $[M]$ matrices are generated for each substructure. $[K_1]$ and $[K_2]$ are both complex since proportional structural damping was applied to both structures, again this damping is arbitrary. $[K_1]$ and $[K_2]$ are of the form $[K] = [K + j\alpha K]$. We next form the impedance matrix for each substructure. The impedance matrix is of the form $[Z] = [K] - \Omega^2[M]$. With the impedance matrix generated for each substructure, the FRF matrix H can be calculated by inverting the impedance matrix. This process is done at each frequency of interest. Now with the FRF matrix for each substructure calculated, we are ready to synthesize the two structures and isolators together to form the structure in Figure 5.1. Referring to the synthesis equation

above, the matrices $[H_{ee}]$, $[H_{ec}]$, $[H_{ce}]$, and $[H_{cc}]$ are formed by combining $[H_1]$ and $[H_2]$ by appropriate partitioning. The partitioning is shown below.

$$[H_{ee}] = \begin{array}{c} \\ i_1 \\ i_2 \\ c_1 \\ c_2 \end{array} \begin{array}{c} \begin{array}{cccc} i_1 & i_2 & c_1 & c_2 \end{array} \\ \left[\begin{array}{cc|cc} H_1(i_1,i_1) & [0] & H_1(i_1,c_1) & [0] \\ [0] & H_2(i_2,i_2) & [0] & H_2(i_2,c_2) \\ \hline H_1(c_1,i_1) & [0] & H_1(c_1,c_1) & [0] \\ [0] & H_2(c_2,i_2) & [0] & H_2(c_2,c_2) \end{array}\right] \end{array}$$

$$[H_{ce}] = \begin{array}{c} \\ c_1 \\ c_2 \end{array} \begin{array}{c} \begin{array}{cccc} i_1 & i_2 & c_1 & c_2 \end{array} \\ \left[\begin{array}{cc|cc} H_1(c_1,i_1) & [0] & H_1(c_1,c_1) & [0] \\ [0] & H_2(c_2,i_2) & [0] & H_2(c_2,c_2) \end{array}\right] \end{array}$$

$$[H_{ec}] = \begin{array}{c} \\ i_1 \\ i_2 \\ c_1 \\ c_2 \end{array} \begin{array}{c} \begin{array}{cc} c_1 & c_2 \end{array} \\ \left[\begin{array}{cc} H_1(i_1,c_1) & [0] \\ [0] & H_2(i_2,c_2) \\ \hline H_1(c_1,c_1) & [0] \\ [0] & H_2(c_2,c_2) \end{array}\right] \end{array} \qquad [H_{cc}] = \begin{array}{c} \\ c_1 \\ c_2 \end{array} \begin{array}{c} \begin{array}{cc} c_1 & c_2 \end{array} \\ \left[\begin{array}{cc} H_1(c_1,c_1) & [0] \\ [0] & H_2(c_2,c_2) \end{array}\right] \end{array}$$

Referring to Figure 5.2, "i1" denotes the set of internal coordinates of structure 1 which include 1, 2, 3, 4, 5, 6, 7, 8, 9, 16, 17, 18, 19, 20, 21, 22, 23, 24, 25, 26, 27, 28 ,29, and 30, "c1" denotes the set of connection coordinates of structure 1 which include 10, 11, 12, 13, 14, 15, 31, 32, 33, 34, 35, and 36, "i2" denotes the set of internal coordinates of structure 2 which include 4, 5, 6, 7, 8, 9, 10, 11, and 12, "c2" denotes the set of connection coordinates of structure 2 which are 1, 2, 3, 13, 14, 15, 16, 17, 18, 19, 20, and 21. With the appropriate partitioning complete, the synthesis of structure 1 to structure 2 can be performed using the indirect coupling relation

$$[H_{ee}]^{*} = [H_{ee}] - [H_{ei}][M]\left[\tilde{Z} + \tilde{H}_{ii}\right]^{-1}[M]^{T}[H_{ie}]. \tag{99}$$

Structure 1 is synthesized to structure 2 by the isolators or load paths described by $[\tilde{Z}]$. $[H_{ee}]^{*}$ is the synthesized FRF relation which is the combination of both structures and isolators. The synthesis is done over the frequency range of interest and the response is plotted in Figure 5.3. The frequency range for this example was 0.1 to 8.0 Hz. Figure 5.4 is the solution from traditional FE calculations for direct comparison. Both plots are identical.

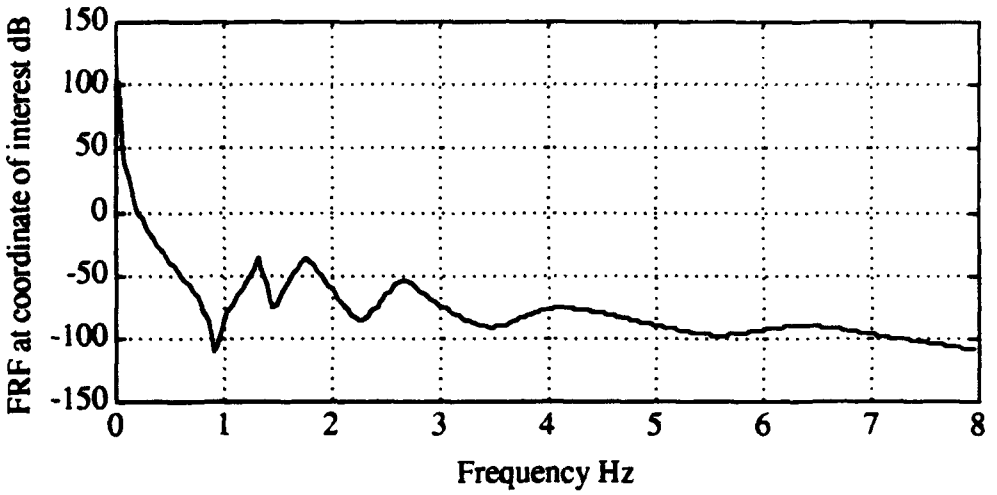


Figure 5.3. Plot of $H_{ee}(8,8)$ for Synthesized System.

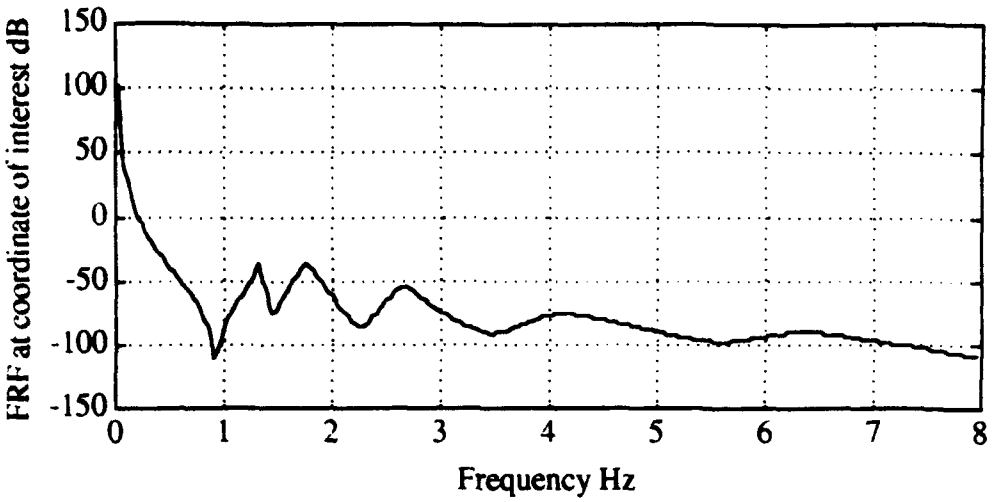


Figure 5.4. Plot of H(8.8) from Traditional FE Calculations.

Figures 5.3 and 5.4 are the plots of the FRF at element (8,8) of the synthesized $[H_{ee}]^{\bullet}$ and traditional FE $[H]$ matrices, respectively. This element (8,8) corresponds to the lateral motion coordinate 8 of Figure 5.1. Notice both plots are identical and show the first five damped natural frequencies. The plots show the magnitude of the response at coordinate 8 due to a unit excitation at varying frequency at coordinate 8. As the frequency of excitation approaches the damped natural frequency, the response approaches infinity.

F. EXAMPLE (6): INDIRECT COUPLING WITH FREQUENCY DEPENDENT ISOLATORS

In this example, we demonstrate the capability of synthesizing components with frequency dependent properties. Specifically, we will repeat the preceding example using isolators that have frequency dependent damping. Consider the following figures. The FRF for the structure shown in Figure 6.1 will be calculated using traditional FE procedures to

compare with the FRF calculated using the synthesize method. The components to be synthesized are shown in Figure 6.2.

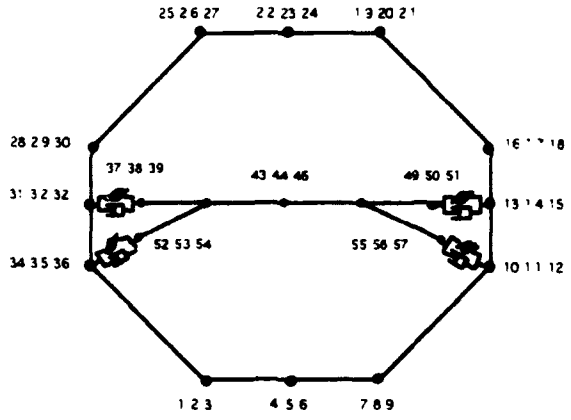


Figure 6.1. Traditional FE Procedures are Used to Calculate FRF for the Combined Hull-Isolator-Cradle Structural System

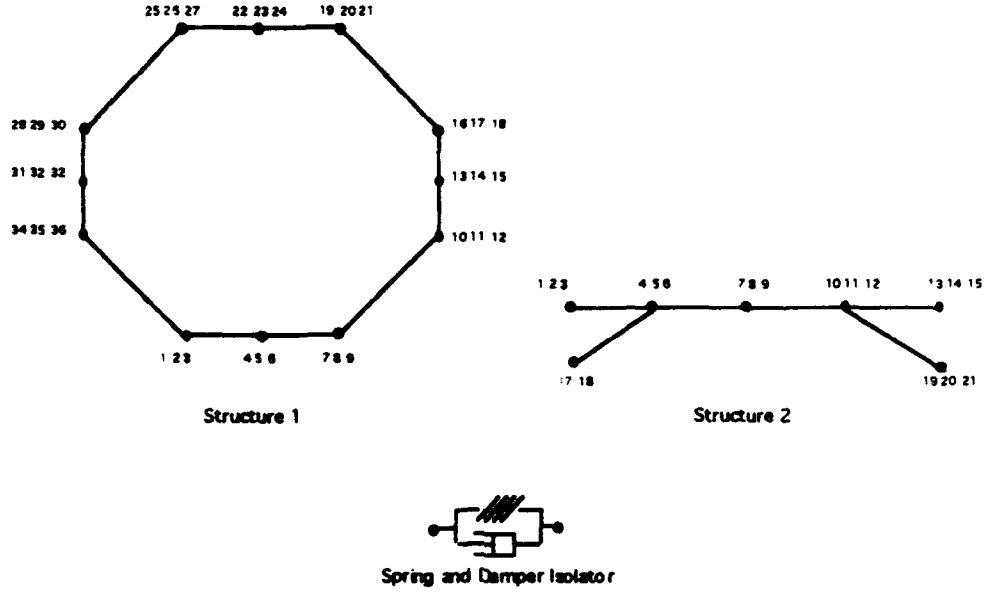


Figure 6.2. Total Hull-Isolator-Cradle System is Synthesized from Components.

In Example (5), the isolators were treated as having proportional viscous damping. This example will use viscous damping which is frequency dependent. The methodology is the

same as in the previous example and will not be repeated here. The discussion here will focus on the only difference which is the damping applied to the isolator. Referring to the general impedance relation $[Z(\Omega)] = [K] - \Omega^2[M] + j\Omega[C]$. $[C]$ is now a function of Ω. The equation is now of the form

$$[Z(\Omega)] = [K] - \Omega^2[M] + j\Omega[C(\Omega)]. \tag{104}$$

The damping applied to the isolator was arbitrarily selected as an exponential decay dependent on frequency. The form of the function used is

$$C = C_o e^{-a\Omega} \tag{3}$$

where $C_o = k = 25$ lb • s/in and a = 0.1.

The damping function is plotted in Figure 6.3.

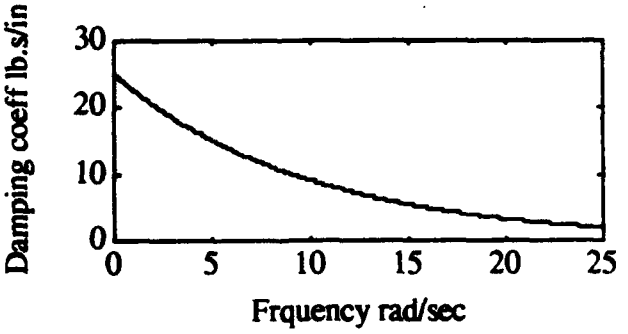


Figure 6.3. Plot of Isolator Damping Versus Frequency

The reduced impedance in this example is now of the form $[\tilde{Z}] = [I](k + j\Omega k e^{-a\Omega})$ and the size of the matrix is (12 x 12).

Structure 1 is coupled to structure 2 with isolators described by $[\tilde{Z}]$. $[H_{rr}]^*$ is the synthesized FRF relation which is the combination of both structures and isolators. The synthesis is done over the frequency range of interest and plotted, Figure 6.4. The frequency range for this example was 0.1 to 8.0 Hz. Figure 6.5 is the solution from traditional FE calculations provided for direct comparison. Both plots are identical.

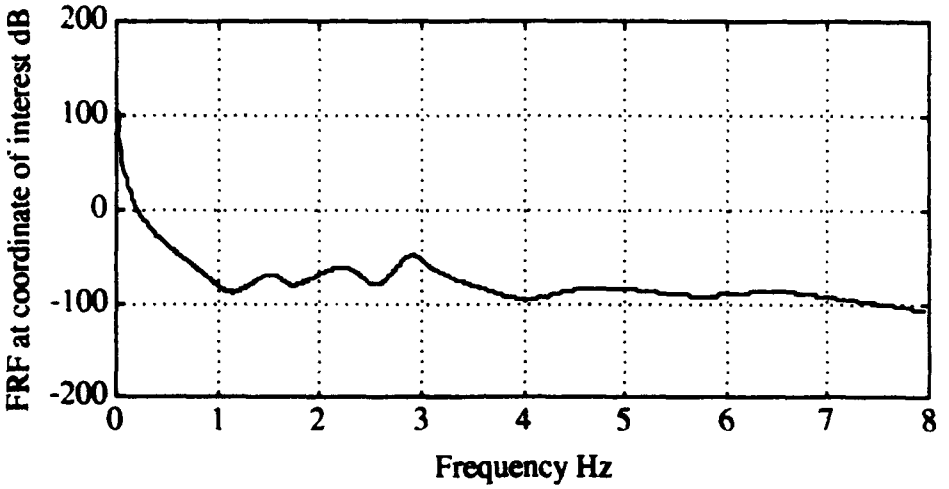


Figure 6.4. Plot of $H_{ee}(8,8)$ Calculated Using the Synthesis Method.

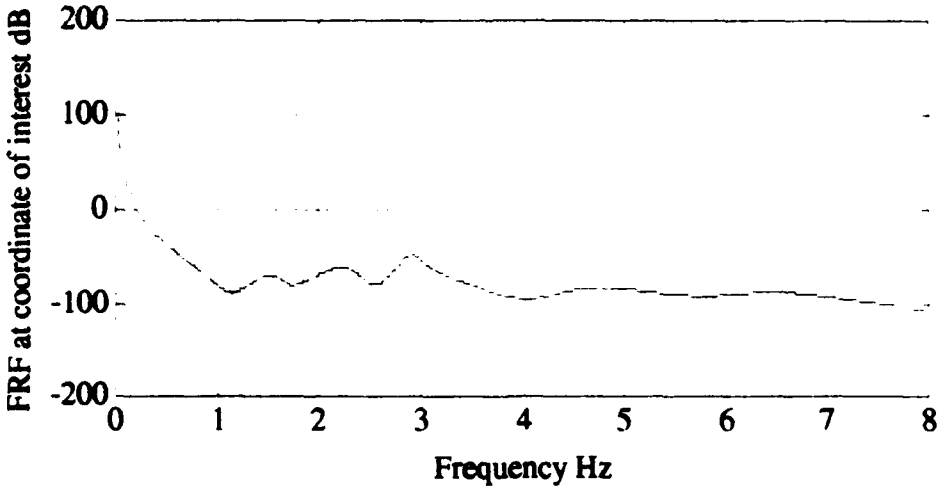


Figure 6.5. Plot of H(8,8) Calculated Using Traditional FE Procedures.

Figures 6.4 and 6.5 are the plots of the FRF at element (8.8) calculated using the synthesized $[H_{ee}]^{\bullet}$ and FE $[H]$ matrices, respectively The FRF element (8.8) corresponds to the lateral motion coordinate 8 of Figure 6.1. Notice both plots are identical and show the first five damped natural frequencies. The plots show the magnitude of the response at coordinate 8 due to a unit excitation at varying frequency at coordinate 8. As the frequency of excitation approaches the damped natural frequency, the response approaches infinity.

A comparison of the compute time required for the synthesis versus traditional FE calculation. The actual computing time and the number of floating point operations (flops) for each method is provided:

FEM direct assembly: time - 25876 sec or 431.3 mins

FLOPS - 1.49×10^9

FRF synthesis: time - 1167 sec or 19.45 mins

FLOPS - 517.2×10^6

This clearly demonstrates that synthesis by FRF is more efficient and well suited for design analysis.

G. EXAMPLE (7): STRESS CALCULATION BY DYNAMIC INDIRECT COUPLING

Consider the structures shown in the following figures. The structure shown in Figure 7.1 will be directly assembled by the finite element method and the peak bending stress frequency response will be calculated in beam element #4 whose location is shown by the dashed line A--A. The same structure will be synthesized using the frequency domain method and the same stress frequency response will be calculated. The FRF results calculated by the synthesis methodology, shown in Figure 7.5, will be compared with that calculated by traditional FEM, Figure 7.6. Again, the structure (specifically, its FRF) as shown in Figure 7.1 will be obtained by synthesizing structure 1 and the modification,

structure 2, as shown in Figure 7.2. Note that a stress frequency response allows the direct calculation of stress due to the application of a force or moment. The equation for determining synthesized stress is shown as equation (105).

$$\{\sigma(\Omega)\}^* = [H_{\sigma e}(\Omega)]^* \{f(\Omega)\}. \tag{105}$$

where the synthesized stress FRF matrix $[H_{\sigma e}(\Omega)]^*$ reflects the total structure, including any modifications or couplings.

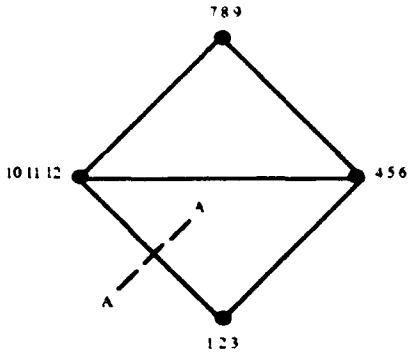


Figure 7.1. Structure Analyzed for Peak Bending Stress

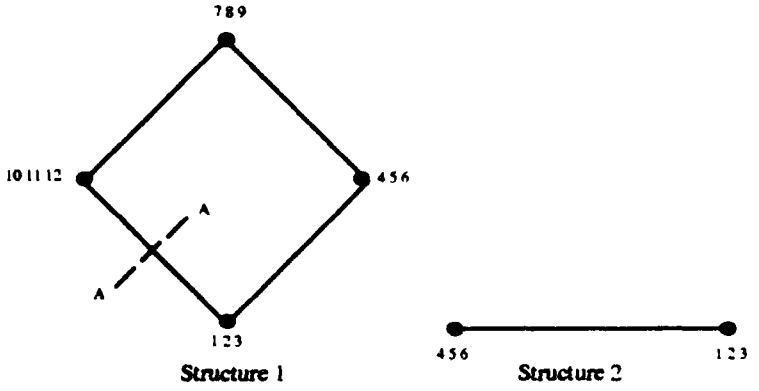


Figure 7.2. Components of Synthesized Structure

Referring to Figure 7.2, structure 1 will be modified by adding the beam, structure 2 at the nodal coordinates 4, 5, 6, 10, 11, and 12. The following beam element data will be used:

Young's Modulus E = 30.0×10^6 psi

Area moment of inertia I = 0.02083 in^4

Cross-sectional area A = 1 in

Weight density WTD = 0.2832 lbf/in^3

Distance from beam center to outer most fiber c = 0.05 in

For this example, damping was not used, but the methodology is able to handle all forms of linear damping as described earlier. The general equation for synthesizing stress information by dynamic indirect coupling/modification is

$$[H_{\sigma e}]^* = [H_{\sigma e}] - [H_{\sigma c}][Z^{-1} + H_{cc}]^{-1}[H_{ce}]. \tag{66}$$

This equation is the first row extracted from the relationship shown as equation (12). Note that the synthesis of stresses can be done at the same time as the synthesis of displacements. We are here demonstrating just the synthesis of stress information.

The first step is to generate the $[K]$ and $[M]$ matrices for structure 1 and the beam element shown in Figure 7.2. Next the impedance matrices are generated for each structure as $[Z(\Omega)] = [K] - \Omega^2[M]$. Since we are modifying structure 1, the FRF matrix $[H]$ is only calculated for structure 1 of Figure 7.2. The complete process as described here is performed over the frequency range of interest. There is basically two sections to this process: (1) the partitioning of the $[H]$ matrix into its required sub matrices for the general synthesis process and (2) the extraction of the information from the $[H]$ matrix and the processing of that information to calculate the stress frequency response.

Referring to Example (4), we partition $[H]$ for structure 1 in the same manner. For the synthesis of stress frequency response only, only the partitions of $[H_{cc}]$ and $[H_{ic}]$ are required. These partition are shown below.

$$[H_{ic}] = \begin{matrix} & c_1 \\ i_1 & \\ c_2 & \end{matrix}\begin{bmatrix} H_1(c_1,i_1) & H_1(c_1,c_1) \\ H_1(c_2,i_1) & H_1(c_2,c_1) \end{bmatrix} \qquad [H_{cc}] = \begin{matrix} & c_1 \\ c_1 & \\ c_2 & \end{matrix}\begin{bmatrix} H_1(c_1,c_1) \\ H_1(c_2,c_1) \end{bmatrix}$$

The internal coordinates "i_1" are 1, 2, 3, 7, 8, and 9. The connection coordinates "c_1" are 4, 5, 6, 10, 11, and 12. The second part of the process requires all or part of the FRF matrix for structure 1, depending on where the external loads are applied. The connection coordinates are required for the synthesis process, as always, and internal coordinates are required if the stress frequency response which is of interest is associated with an element whose nodal coordinates are internal coordinates, i.e. they are not directly associated with the synthesis. In this example, all coordinates are used for the stress information. We apply a unit load at each coordinate using the following equation,

$$\{x\}^i = [H]\{f_i\}. \tag{106}$$

where i is an element of the required coordinates. This equation is interpreted as the displacements at the structural system coordinates due to the unit load at the desired coordinates of interest, which is the combination of connection coordinates and any internal coordinates desired. $\{x\}^i$ is the i'th column of $[H]$ when using unit forces. Using the i'th column of $[H]$, we extract the elements corresponding to the beam element that stress information is desired for, getting a partitioned form of the i'th column. The complete reduced form of the $[H]$ matrix is $[H(bc,dc)]_{Reduced}$, where bc are the coordinates of the beam of interest and dc is the set of required coordinates we wish to keep. In our example,

the set of beam coordinates, "bc" is 1, 2, 3, 10, 11, and 12 and the set of required coordinates, "dc" is 1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, and 12. $[H]_{Reduced}$ is a matrix of size (6 x 12). Each column of $[H]_{Reduced}$ is in the global coordinate system and needs to be transformed to local coordinates by using the following relation

$$\{H^i\}_{Reduced_{local}} = [T]\{H^i\}_{Reduced_{global}} \tag{107}$$

where the transformation matrix $[T]$ is

$$[T] = \begin{bmatrix} \cos\alpha & \sin\alpha & 0 & 0 & 0 & 0 \\ -\sin\alpha & \cos\alpha & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & \cos\alpha & \sin\alpha & 0 \\ 0 & 0 & 0 & -\sin\alpha & \cos\alpha & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}.$$

Now with the H transformed to the local coordinate system, we can get the nodal forces by the following relation

$$\{f^i\} = [k_{el}]\{H^i\}_{Reduced_{local}} \tag{108}$$

where $[k_{el}]$ is the elemental stiffness matrix in the local coordinates system. We are ready to solve for the FRF stress, first combining all the column vectors $\{f^i\}$ into a nodal force matrix $[F]$ and then multiplying it by the moment equation to solve for the FRF peak internal bending stress of the beam element.

The FRF stress equation is

$$\{H_\sigma\} = \frac{c}{I}\{M_{eq}\}[F] \ . \tag{109}$$

The $\{H_\sigma\}$ is a row vector size (1 x 12), since we used all the coordinates as our set of desired coordinates. Noting that $\{M_{eq}\}$ is determined from equilibrium for the element in question, and here we provide the internal bending moment. The derivation is shown using Figures 7.3 and 7.4.

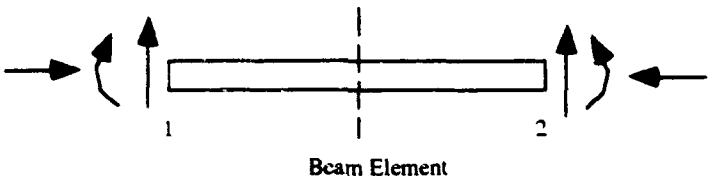


Figure 7.3. Beam Element for Stress Calculation

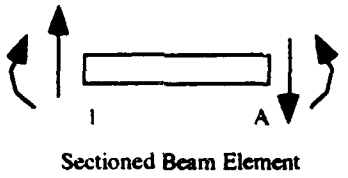


Figure 7.4. Beam Section Cut at the Midpoint

Consider the beam element in Figure 7.3, The moment is to be calculated at the midpoint of the beam. First the beam is cut , Figure 7.4 and the moment at A is solved for:

$$M_A = M_1 + V_1\frac{l}{2} \tag{110}$$

The moment equation in vector form is shown as equation (111).

$$\{M_{eq}\} = \{M_A\} = \{0 \quad l/2 \quad 1 \quad 0 \quad 0 \quad 0\} \tag{111}$$

Noting that the nodal force of the beam element is

$$\{F^i\} = \begin{Bmatrix} A_1 \\ V_1 \\ M_1 \\ A_2 \\ V_2 \\ M_2 \end{Bmatrix} \tag{112}$$

where "A" indicates an axial force, "V" a shear force, and "M," a moment. The internal bending stress frequency response component is determined by equation (113).

$$H_\sigma^i = \frac{c}{I}\{0 \quad l/2 \quad 1 \quad 0 \quad 0 \quad 0\} \begin{Bmatrix} A_1 \\ V_1 \\ M_1 \\ A_2 \\ V_2 \\ M_2 \end{Bmatrix} \tag{113}$$

H_σ^i is evaluated over all the chosen required coordinates to form $\{H_\sigma\}$, which in our example is a row vector size (1 x 12). If more then one beam element is used for stress calculations then $[H_\sigma]$ could be of the size: (number of beam elements) x (number of desired coordinates). With $[H_\sigma]$ generated, we can now partition it into the required sub-partitions for synthesis. The partitions required are $[H_{\sigma c}]$ and $[H_{\sigma e}]$ and are

$$[H_{\sigma c}] = nb\left[H_{\sigma}(nb,c1)\right] \qquad [H_{\sigma c}] = nb\left[H_{\sigma}(nb,i1) \quad H_{\sigma}(nb,c1)\right]$$

The beam element set is indicated by "nb". In this example "nb" could have been 1, 2, 3, and 4 since there are four beam elements in the substructure to be modified. Beam element 4 was chosen as the beam to calculate FRF stress information so "nb" is 4 and the sizes of the matrices are (1 x 6) and (1 x 12) respectively. To get the stress information for beam five, the synthesis method of direct coupling must be used. With the appropriate partitioning completed the synthesis can now be performed. $[H_{\sigma e}]$ is the modified FRF stress relation which is the combination of structure 1 and the added element structure 2 of Figure 7.2. The synthesis is performed over the frequency range of interest and plotted in figure 7.5. The frequency range for this example is 0.3 to 102 Hz. Figure 7.6 is the solution from the traditional FE calculation for direct comparison of the two solutions. Both plots are identical.

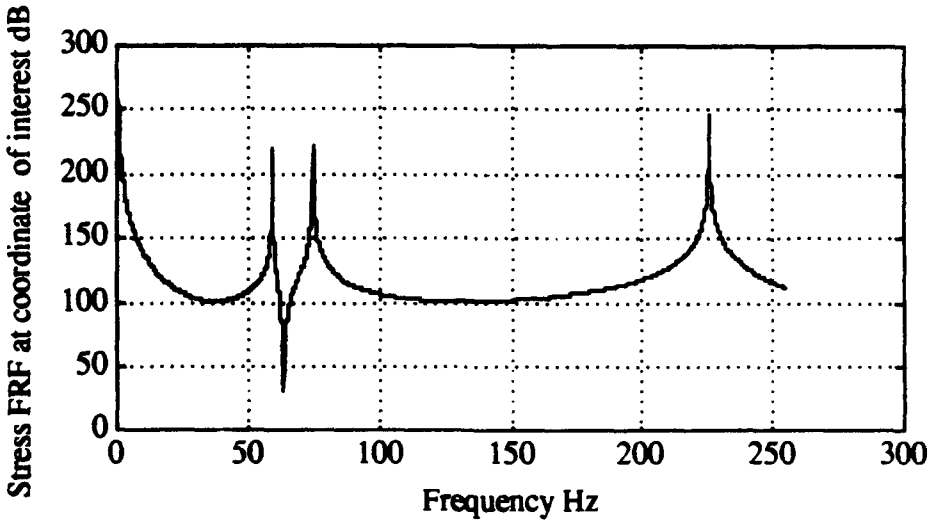


Figure 7.5. Plot of Synthesized FRF Stress Element $H_{\sigma e}(1,9)$

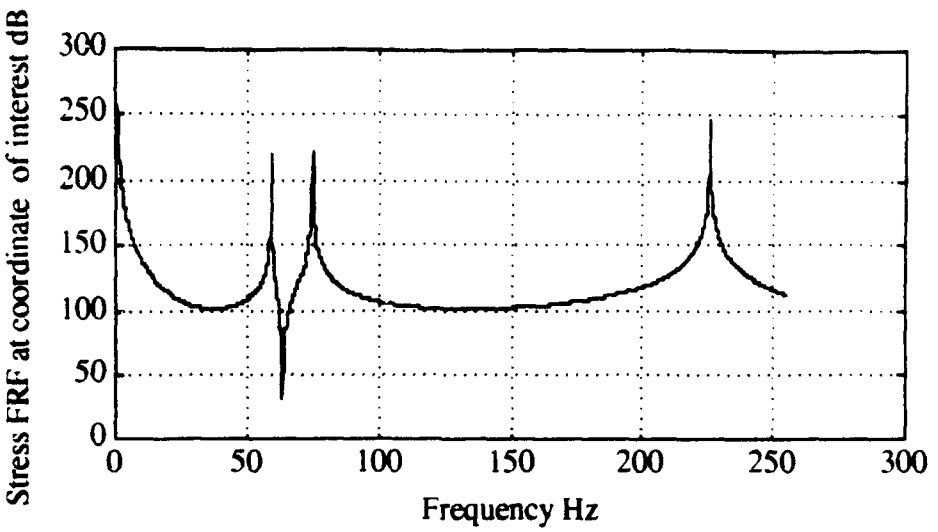


Figure 7.6. Plot of H_{n} (1,6) Calculated Using Traditional FE Procedures.

Figures 7.5 and 7.6 are the plots of the FRF stress corresponding to beam element four of Figure 7.1. These plots represent the stress amplitude in beam element four due to a unit force applied at coordinate 6. Both plots are identical. Note that the frequency of peak response is slightly lower than the undamped natural frequency.

H. EXAMPLE (8): DYNAMIC DIRECT COUPLING USING MODAL REPRESENTATION OF FRF

Consider the structures shown in the following figures. The structure shown in Figure 8.1 will be directly assembled by the finite element method in order to compare the frequency response calculated by traditional FE methods and the solution obtained by synthesizing structure 1 and structure 2 as shown in Figure 8.2. This example will show three results, the first being the solution from synthesis using the modal representation of the frequency response, the second being direct assembly using FE and the modal representation and thirdly, direct assembly using FE where frequency response is calculated by the inverse of the impedance matrix.

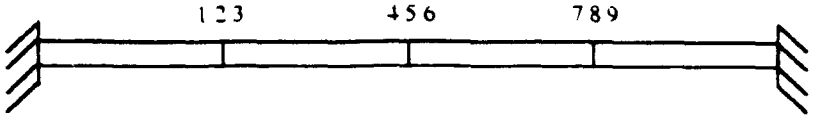


Figure 8.1. Structure Analyzed Using Traditional FE

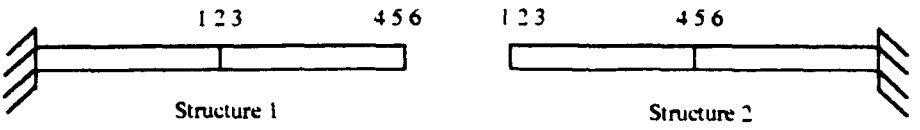


Figure 8.2. Structures to be Synthesized Using Modal Representation

Referring to Figure 8.2, structures 1 and 2 will be synthesized by direct coupling using connection coordinates 4, 5, and 6 of structure 1 and connection coordinates 1, 2, and 3 of structure 2. One internal coordinate will be kept in this synthesis process to show that the frequency response for a specific coordinate can be synthesized using just the connection coordinates and any internal coordinates that might be of interest. This example will use the internal coordinate "2" of structure 1 as the coordinate of interest. The information desired in this example is the frequency response at coordinate 2 due to a unit harmonic load at coordinate 6. Note that when structure 1 and structure 2 are synthesized, the coordinate numbering becomes the same as depicted in Figure 8.1. The following beam element data was used:

Young's Modulus E = 30.0×10^6 psi

Area moment of inertia I = 0.02083 in^4

Cross-sectional area A = 1 in

Weight density WTD = 0.2832 lbf/in^3

Structural proportional damping was not used, but the methodology will handle all forms of damping discussed earlier. The frequency response matrix $[H]$ can be generated by two methods. The first is by the relationship

$$[H(\Omega)] = [Z(\Omega)]^{-1} \tag{14}$$

where $[Z(\Omega)] = [K] - \Omega^2[M] + j\Omega[C]$.

The second method is by matrix modal representation. The relationship is

$$[H(\Omega)] = [\Phi]\begin{bmatrix} \ddots & & \\ & \frac{1}{\omega_r^2 - \Omega^2} & \\ & & \ddots \end{bmatrix}[\Phi]^T \tag{42}$$

where $[\Phi]$ is the set of eigenvectors or mode shapes, and the middle term is the diagonal matrix of the natural frequencies or eigenvalues less the frequency of interest. This relationship can also be expressed in terms of individual elements of the frequency response function by equation (114)

$$H_{ij} = \sum_{r=1}^{n\ modes} \frac{\phi_i^r \phi_j^r}{\omega_r^2 - \Omega^2}, \tag{114}$$

which allows the calculation of specific frequency response of interest without having to generate the complete FRF.

The general synthesis equation for dynamic direct coupling is

$$[H_{ee}]^* = [H_{ee}] - [H_{ec}][M]\left[\tilde{H}_{cc}\right]^{-1}[M]^T[H_{ce}] \ . \tag{84}$$

This equation is written in terms of all the coordinates. This example concerns the synthesis of a single FRF matrix element involving one coordinate. In general, the synthesis process requires FRF information for all connection coordinates, and FRF information for any internal coordinates of interest. Rewriting the general equation for this specific example, the equation becomes

$$[H_{2c}]^* = [H_{2c}] - [H_{2c}][M]\left[\tilde{H}_{cc}\right]^{-1}[M]^T[H_{cc}] \tag{115}$$

where the subscript "2" signifies the internal coordinate "2" of structure 1 and the subscript "c" signifies the set of connection coordinates of both structures. In this equation, [M] is the boolean mapping matrix which is used to establish the connectivity between the two substructures for synthesis. The mapping matrix is determined by the connectivity i.e. what is connected to what and by imposing the equilibrium and compatibility relations associated with each pair of coordinates. We can define the mapping matrix by $\{f_c\} = [M]\left\{\tilde{f}_c\right\}$. Where $\{f_c\}$ is a vector of all the connection coordinates of both structures and $\left\{\tilde{f}_c\right\}$ is the arbitrarily selected independent subset of the connection coordinates relating to one of the substructures. We have selected the connection coordinates of structure 1 as the arbitrary subset of connection coordinates. The mapping matrix [M] is a matrix of size (6 x 3) and is depicted as:

$$[M] = \begin{bmatrix} 1 & & \\ & 1 & \\ & & 1 \\ \hline -1 & & \\ & -1 & \\ & & -1 \end{bmatrix}$$

The FRF matrix $[H]$ for both substructures 1 and 2 is calculated using the matrix modal representation relation discussed earlier. The important point here is that we are not using all the coordinates. The coordinates used from substructure 1 are 2, 4, 5, and 6. the first coordinate is the coordinate of interest and the rest are connection coordinates which must be used. The coordinates used from substructure 2 are just the required connection coordinates 1, 2, and 3. All six mode shapes for each coordinate are kept for the calculation of the FRF matrix. The FRF matrix $[H_1]$ is calculated by using the appropriate partitioning of the modal matrix $[\Phi_1]$. The diagram of the relation on the next page is showing the coordinates kept and the number of modes.

$$[H_1] = \begin{matrix} 2 \\ 4 \\ 5 \\ 6 \end{matrix}\overset{6}{\left[\ \Phi_1\ \right]}\overset{6}{\left[\ \begin{matrix} \frac{1}{\omega_r^2 - \Omega^2} & & \\ & \ddots & \\ & & \end{matrix}\ \right]}\overset{2\ 4\ 5\ 6}{\left[\ \Phi_1^T\ \right]}$$

The size of $[H_1]$ is now a (4 x 4) matrix which contains all the necessary information. This also shows a significant computational advantage because the size has been reduced from a (6 x 6) to a (4 x 4) matrix which requires less computational time to manipulate the matrix. $[H_2]$ is generated in the same manner. We will use all six mode shapes for each coordinate kept of substructure 2. The required coordinates are the connection coordinates 1, 2, and 3. The diagram of the relation is given on the next page showing the coordinates kept and the number of modes.

$$[H_2] = \begin{matrix} \\ 1 \\ 2 \\ 3 \end{matrix} \begin{matrix} \begin{matrix} 6 \end{matrix} \\ \begin{bmatrix} \\ \Phi_2 \\ \\ \end{bmatrix} \end{matrix} \begin{matrix} \begin{matrix} 6 \end{matrix} \\ \begin{bmatrix} \\ \frac{1}{\omega_r^2 - \Omega^2} \\ \\ \end{bmatrix} \end{matrix} \begin{matrix} \begin{matrix} 1 & 2 & 3 \end{matrix} \\ \begin{bmatrix} \\ \Phi_2 \\ \\ \end{bmatrix} \end{matrix}$$

Now with h1 and h2 generated, the two substructures can be synthesized. Referring to the example-specific synthesis equation above, the matrices $[H_{2c}]$ and $[H_{cc}]$ are formed by combining h1 and h2 by appropriate partitioning. The partitioning is shown on the next page.

$$[H_{2c}] = \begin{matrix} & c_1 & c_2 \\ i_1 & \end{matrix} \begin{bmatrix} H_1(i_1,c_1) & [0] \end{bmatrix} \qquad [H_{cc}] = \begin{matrix} & c_1 & c_2 \\ c_1 & \\ c_2 & \end{matrix} \begin{bmatrix} H_1(c_1,c_1) & [0] \\ [0] & H_2(c_2,c_2) \end{bmatrix}$$

Referring to Figure 8.2, "i1" is the set of internal coordinates for substructure 1. Since coordinate 2 is the only coordinate of interest, the set of internal coordinates is just coordinate 2. The set of connection coordinates "c1" consists of 3, 4, and 5 and "c2" consists of 1, 2, and 3. With the appropriate partitioning complete, the two structures are synthesized together, to form the structure in Figure 8.1, using the case specific form of the direct coupling relation

$$[H_{2c}]^* = [H_{2c}] - [H_{2c}][M]\left[\tilde{H}_{cc}\right]^{-1}[M]^T[H_{cc}]. \qquad (115)$$

$[H_{2c}]^*$ is the synthesized FRF by modal representation relation which is the combination of both structures. The synthesis is done over the frequency range of interest and plotted in Figure 8.3. The frequency range for this example was 0.1 to 80.0 Hz. Figure 8.4 is the

solution from traditional FE calculations using the inverse of the impedance matrix to calculate the FRF and Figure 8.5 is the solution from traditional FE calculations using the modal representation to calculate the FRF. Figures 8.4 and 8.5 are included for direct comparison. All three plots are identical.

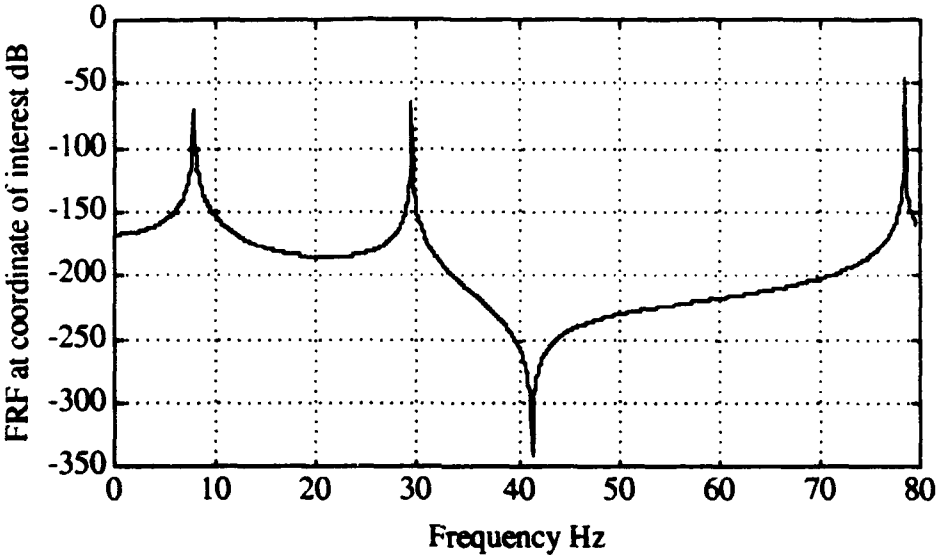


Figure 8.3. Plot of Synthesized $H_{2c}(1,3)$

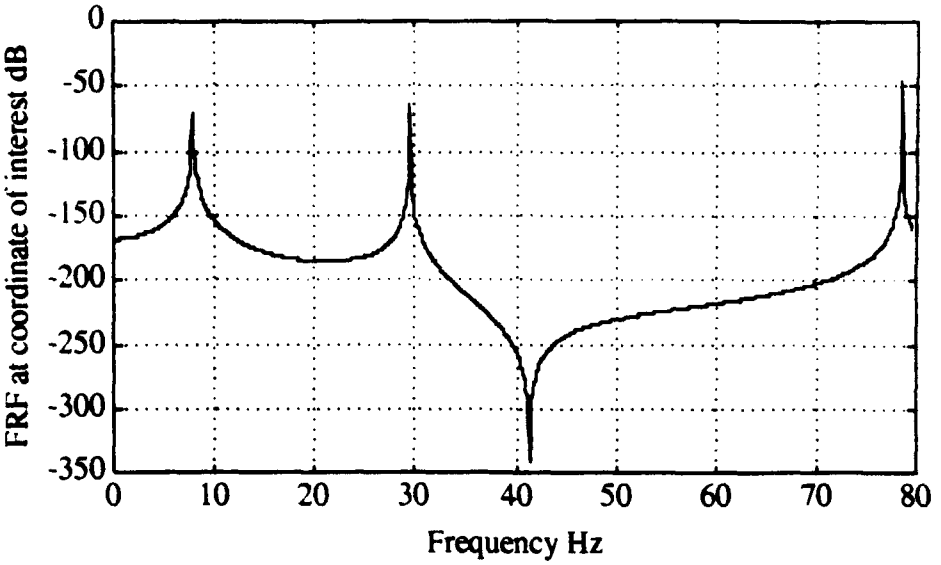


Figure 8.4. Plot of H(2,6) from Traditional FE Calculations

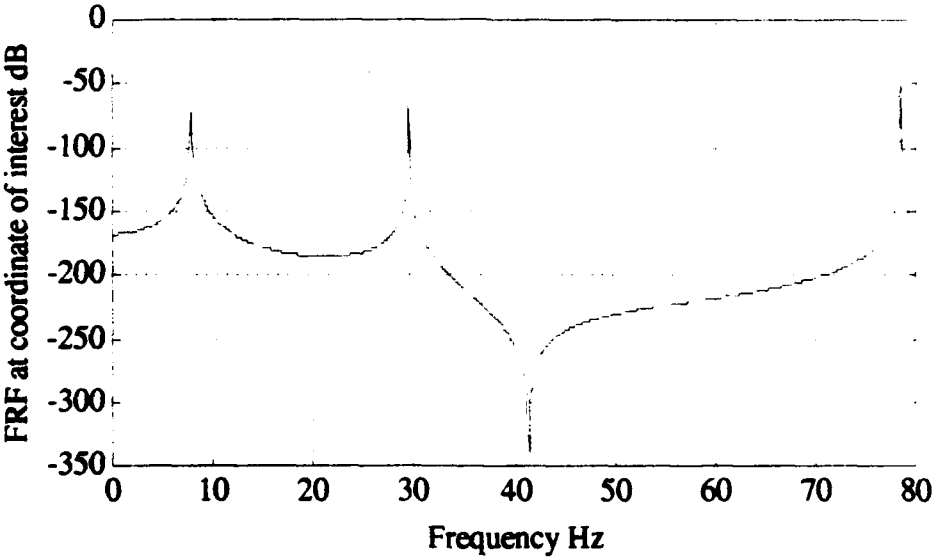


Figure 8.5. Plot of H(1,4) from Traditional FE Calculations Using Modal Representation

Figures 8.3, 8.4 and 8.5 are the plots of the FRF at element (1,3), (2,6), and (1,4) respectively. These elements corresponds to the lateral motion coordinate 2 of Figure 8.1.

A special note here is that the element (2,6) of the FRF generated by FEM is the response at coordinate 2, which corresponds to the element (1,3) of the synthesized FRF generated by the direct coupling relation using modal representation, and element (1,4) of the FRF generated by traditional FE using modal representation. The reason for this is because of the partitioning and the coordinates used in the calculation. Care is required here to ensure the coordinate of interest is actually being used. The plots show the magnitude of the response at coordinate 2 due to a unit excitation at varying frequency at coordinate 6. As the frequency of excitation approaches the natural frequency of response, the response approaches infinity.

Figure 8.6 is the plot of the determinant of $\tilde{H}_{cc}$ which shows the natural frequencies of the synthesized structure. The frequencies where the plot crosses the axis or equivalently, the frequencies for which the $\det\left[\tilde{H}_{cc}\right] = 0$ correspond to the natural frequencies of the synthesized structure. This information is important because it gives the designer a starting point on deciding how many modes to keep in the modeling of the system and the frequency bandwidth over which to perform the synthesis. Reducing the number of retained modes will decrease the computational cost and the computer time required to analyze a given design. The number of modes required to accurately model a given structure is case specific.

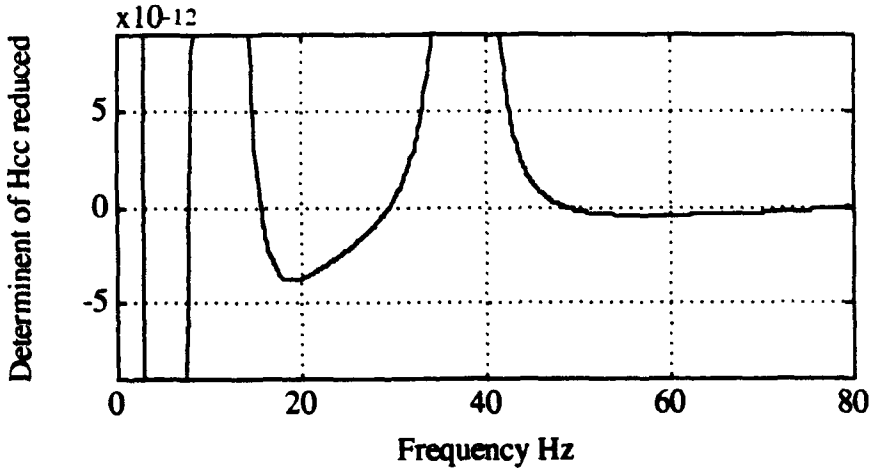


Figure 8.6. Plot of the Determinant of $\tilde{H}_{cc}$ (Plotted over Reduced Bandwidth)

V. CONCLUSIONS AND RECOMMENDATIONS

The most important conclusion from this study is that the analysis and re-analysis of structural systems is performed most efficiently by working in the frequency domain. It was shown in example (6) that synthesis of complex structures was approximately 22 times faster than the traditional FE methods. The large increase in efficiency means that rapid analysis and re analysis of structures can be performed. Large scale structural analysis can now be looked at in man hours where analysis by traditional FE methods is in man days.

Structural synthesis in the frequency domain provides for an arbitrary order model reduction that requires only the coordinates involved in the synthesis and any other coordinates that might be of interest. The solution to the reduced model is exact. This is a significant point because a 10,000 degree of freedom model can be reduced to a system of tens or hundreds of degrees of freedom, significantly improving the computational efficiency.

The frequency response theory allows for the direct synthesis of response information of any kind. Using a generalized definition of frequency response, displacement, velocity, acceleration, stress, and strain information may be directly synthesized. based on this generalization, the theory is an ideal means for doing static and dynamic design re-analysis. Static problems are treated as the zero frequency case.

The frequency domain structural synthesis theory allows for any combination of substructure coupling and structural modification to be performed, either simultaneously or sequentially.

Recommendations to further utilize and demonstrate the theory of frequency domain structural synthesis is first, to write computer code that will interface with existing finite element codes, for example NASTRANS (MSC, Corp.) or IDEAS (SDRC, Corp.) to synthesize substructures in three dimensions using plate, shell, and beam elements with six degrees of freedom per node which allows for out of plane analysis. Using the combination of plate, shell, and beam elements will more closely approximate actual structures. Second, build a scaled prototype of a submarine and equipment cradle and compare the theoretical results with experimental results.

APPENDIX A MATLAB CODE FOR EXAMPLE ONE

```
clg
clear

%                          unifineel
%
% This program will calculate the eigenvalues,eigenvectors
% natural frequencies, and frequency response function matrix
% for a three degree of freedom at each node element.
% The system is modeled with beam elements that are
% aligned in the same plane but at any angle (2-D).
%          •--------•
%          |        |
%          |        |
%          •--------•
%          |        |
%          |        |
%          •--------•
% This program works for a beam element modeled with six general
% coordinates and thus six DOF.
%          (|-•--------•-|)
%
% the user must enter the following data to meet the beam configuration
% (E) youngs modulus psi
% (I) area moment of inertia in^4
% (WTD) weight density lbf/in^3
% (A) cross sectional area in^2
% conductivity [ the node connection mapping ]
% node coordinates [ cartesian coordinates for each node ]
% (bb) proportional damping constant
clear;
% call the data file
beam2
%
% start the program clock and flops to determine program running
% time and floating point calculations
t0=clock;
flops(0);
%
% calculate the number of beam elements
a=size(con);
numel=a(1);
%
% calculate the number of beam elements proportionally damped
aa=size(dcon);
```

```
numel_damp=aa(1);
%
% calculate the number of nodes.
b=size(coord);
nodes=b(1);
%
% convert the coordinates in to the correct units (in.)
coord=coord*12;
%
% calculate the beam element lengths and beam angles
% in radians
for i=1:numel
IC=con(i,1);
ID=con(i,2);
l(1,i)=sqrt((coord(ID,1)-coord(IC,1))^2+(coord(ID,2)-coord(IC,2))^2);
DX(i)=coord(ID,1)-coord(IC,1);
DY(i)=coord(ID,2)-coord(IC,2);
  if DX(i)>=0 & DY(i)>=0;
     t(1,i)=acos(DX(i)/l(1,i));
  elseif DX(i)<0 & DY(i)>=0;
     t(1,i)=acos(DY(i)/l(1,i))+pi/2;
  elseif DX(i)<0 & DY(i)<=0;
     t(1,i)=acos(abs(DX(i))/l(1,i))+pi;
  else
     t(1,i)=acos(abs(DY(i))/l(1,i))+(3*pi/2);
  end;
end;
%
% call trig function
[c,s]=ftrig(t,numel);
%
% calculate radius of gyration
for i=1:numel
r(1,i)=sqrt(I(i)/A(i));
end;
%
% create the global matrix which is all zeroes.
%
kg=[zeros(nodes*3,nodes*3)];
mg=[zeros(nodes*3,nodes*3)];
%
% assemble the elemental matrices to the global matrix.
%
for i=1:numel
[kel,mel]=felement6(l(i),WTD(i),I(i),E(i),A(i),r(i),c(i),s(i));
%
v=con(i,1);
w=con(i,2);
%
kg(3*v-2:3*v,3*v-2:3*v) = kg(3*v-2:3*v,3*v-2:3*v) + kel(1:3,1:3);
kg(3*v-2:3*v,3*w-2:3*w) = kg(3*v-2:3*v,3*w-2:3*w) + kel(1:3,4:6);
```

```
kg(3*w-2:3*w,3*v-2:3*v) = kg(3*w-2:3*w,3*v-2:3*v) + kel(4:6,1:3);
kg(3*w-2:3*w,3*w-2:3*w) = kg(3*w-2:3*w,3*w-2:3*w) + kel(4:6,4:6);
%
mg(3*v-2:3*v,3*v-2:3*v) = mg(3*v-2:3*v,3*v-2:3*v) + mel(1:3,1:3);
mg(3*v-2:3*v,3*w-2:3*w) = mg(3*v-2:3*v,3*w-2:3*w) + mel(1:3,4:6);
mg(3*w-2:3*w,3*v-2:3*v) = mg(3*w-2:3*w,3*v-2:3*v) + mel(4:6,1:3);
mg(3*w-2:3*w,3*w-2:3*w) = mg(3*w-2:3*w,3*w-2:3*w) + mel(4:6,4:6);
end
%
%
% apply structural prop. damping to the k matrix and set
% global k matrix to equal damped matrix
%
kgd=kg;
for i=1:numel_damp
u=dcon(i,1);
[kel]=felement6(l(u),WTD(u),I(u),E(u),A(u),r(u),c(u),s(u));
%
v=dcon(i,1);
w=dcon(i,2);
%
kgd(3*v-2:3*v,3*v-2:3*v) = kgd(3*v-2:3*v,3*v-2:3*v) + j*bb*kel(1:3,1:3);
kgd(3*v-2:3*v,3*w-2:3*w) = kgd(3*v-2:3*v,3*w-2:3*w) + j*bb*kel(1:3,4:6);
kgd(3*w-2:3*w,3*v-2:3*v) = kgd(3*w-2:3*w,3*v-2:3*v) + j*bb*kel(4:6,1:3);
kgd(3*w-2:3*w,3*w-2:3*w) = kgd(3*w-2:3*w,3*w-2:3*w) + j*bb*kel(4:6,4:6);
%
end
%
% apply the boundary conditions
% the user must adjust the global matrix to meet the boundary conditions
%
% to delete rows
kg([BC],:) = [ ];
kgd([BC],:) = [ ];
mg([BC],:) = [ ];
% to delete columns
kg(:,[BC]) = [ ];
kgd(:,[BC]) = [ ];
mg(:,[BC]) = [ ];
%
% call the function and calculate eigenvectors and the eigenvalues
% which are the mode shapes and undamped natural frequency in
(rad/sec)^2
%
[lambda,phi]=fgmodes(kg,mg);
% this now converts the eigenvalues to nat frequency in (rad/sec) and
% and hertz(1/sec)
%
omega = sqrt(lambda);
freq = omega/(2*pi);
%
```

```
% construct the frequency response plot over the frequencies
% of interest
count=0;
for Comega=2:.5:375
count=count+1;
Z=kgd-Comega^2*mg;
H=inv(Z);
%
% this determines the coordinate of interest to plot
HH(count)=H(5,5);
end;
% end the program clock and flops
etime(clock,t0),flops
Comega=2:.5:375;
Freq=Comega/(2*pi);
plot(Freq,20*log(HH)),grid
xlabel(' Frequency Hz ')
ylabel('FRF at coordinate of interest dB')
%
% END


%             beam2
% This is the data for the full structure |--:--:--|
%
%
% The data will be in the form of
% - (E) youngs modulus psi
% - (I) area moment of inertia in^4
% - (WTD) weight density lbf/in^3
% - (A) cross sectional area in^2
% - conductivity [ the node connection mapping ]
% - node coordinates [ cartesian coords for each node in ft. ]
%                      the main program will convert to in..
% - (bb) structural proportional damping constant
%
E=[30 30 30 ]*1e6;
I=[.1666 .1666 .1666 ]*1e-3;
A=[.2 .2 .2 ];
WTD=[.2832 .2832 .2832 ];
bb=.02;
% nodal connectivity
con=[1,2;
     2,3;
     3,4];
% nodal damping connectivity
dcon=[1,2;
      3,4];
% nodal cartesian coordinates
coord=[0,0;
       2,0;
```

```
       4,0;
       6,0];
% boundary conditions
BC=[1 2 3 10 11 12];


%              beam1
% This data handles all three substructures for synthesis
%  |---- ,  ---- , ----|
%
% The data will be in the form of
% - (E) youngs modulus psi
% - (I) area moment of inertia in^4
% - (WTD) weight density lbf/in^3
% - (A) cross sectional area in^2
% - conductivity [ the node connection mapping ]
% - node coordinates [ cartesian coords for each node in ft. ]
%                      the main program will convert to in..
% - (bb) structural proportional damping constant
E=[30 ]*1e6;
I=[.1666 ]*1e-3;
A=[.2 ];
WTD=[.2832 ];
bb=.02;
% nodal connectivity
con=[1,2];
% nodal damping connectivity
dcon=[1,2];
% nodal cartesian coordinates
coord=[0,0;
       2,0];
% Boundary conditions
% structure 1
%BC=[1 2 3];
% structure 2
BC=[4 5 6];
% impedence z
%BC=[ ];


clear;
clg;
% This is example 1 which demonstrates dynamic indirect coupling.
% Two structures will be synthesized together by way of the new load
% path z to form one structure that is restrained at both ends
%
%        |-----    -----   -----|
%           1        z       2
%
%    Hee = Hee* - Hec * inv( inv(z) + Hcc) * Hce
%
```

```
% Load data from running "unifineel program" for each substructure,
% the K and M matrix for each structure is saved
load c1.mat     % k1,m1 is stored here
load c2.mat    % k2,m2 is stored here
load c3.mat     % kz,mz is stored here
%
%  We need to create a single FRF matrix representing
%  both substructures in the form:
%
%       [hee] =  [ h(i,i)   h(i,c) ]
%                [ h(c,i)   h(c,c) ]
%
%  So we create arrays containing the DOF numbers of our original
%  models which correspond to the "c" and "i" coordinates for
%  each substructure.
%
% call the synthesis data file in now which contains the
% internal coordinates and connection coordinates for each sub
% structure.
% i1= internal coords of sub structure 1
% i2= internal coords of sub structure 2
% c1= connection coords of sub structure 1
% c2= connection coords of sub structure 2
%
FRF_INDIR_DATA
%
t0=clock;
flops(0);
count=0;
for Comega=2:.5:375
count=count+1;
%
%           Form Frequency Response Models for Each Substructure
%           ---- --------- -------- ------ --- ---- ------------
z1=k1-Comega^2*m1;
z2=k2-Comega^2*m2;
z=k3-Comega^2*m3;
h1=inv(z1);
h2=inv(z2);
a=size(i1);
b=size(c1);
c=size(i2);
d=size(c2);
%
aa=a(2);
bb=b(2);
cc=c(2);
dd=d(2);
%
%   Remember, we are trying to calculate the following:
%
```

```
%   hee* = hee - hec * inv(inv(z) + hcc  * hce
%
%  So we need to assemble [hee], [hec], [hce] and [hcc] using the
%  the coordinate sets we just defined.
%  These matrices contain the FRF data for both substructures
%  prior to coupling, i.e the pre-synthesis FRF data.
%
%
%                    Coordinate Partitioning
%                    ---------- ------------
%  Build up uncoupled FRF matrix and sub-partitions:
%
hee = [h1(i1,i1)  zeros(aa,cc)   h1(i1,c1)  zeros(aa,dd);
      zeros(cc,aa)   h2(i2,i2)  zeros(cc,bb)  h2(i2,c2);
       h1(c1,i1)   zeros(bb,cc)   h1(c1,c1)  zeros(bb,dd);
       zeros(dd,aa)  h2(c2,i2)  zeros(dd,bb)  h2(c2,c2)];
%
hec = [h1(i1,c1)  zeros(aa,dd);
       zeros(cc,bb)  h2(i2,c2);
       h1(c1,c1)   zeros(bb,dd);
       zeros(dd,bb)  h2(c2,c2)];
%
hcc =  [h1(c1,c1) zeros(bb,dd);
        zeros(dd,bb)  h2(c2,c2)];
%
hce =  [h1(c1,i1)   zeros(bb,cc)   h1(c1,c1)  zeros(bb,dd);
       zeros(dd,aa)  h2(c2,i2)  zeros(dd,bb)  h2(c2,c2)];
%
%  We can now perform the synthesis:
%
heestar = hee - hec * inv(inv(z) + hcc) * hce;
%
% remove the redundant information
heestar = heestar(1:nodes*3,1:nodes*3);
%
% look at the coordinate of interest
HH(count)=heestar(5,5);
end;
etime(clock,t0),flops
Comega=2:.5:375;
Freq=Comega/(2*pi);
plot(Freq,20*log(HH)),grid
xlabel(' Frequency Hz ')
ylabel(' FRF at coordinate of interest db ')

%                     FRF_INDIR_DATA
%
%
% The following data will be provided by this file
% for the synthesis program.
%
```

```
% i1- internal coords sub structure 1
% i2- internal coords sub structure 2
% c1- connection coords sub structure 1
% c2- connection coords sub structure 2
%
i1=[ ];
c1=[1 2 3];
i2=[ ];
c2=[1 2 3];
%
% enter the number of unrestrained nodes of the synthesized
% structure
nodes = 2;
```

APPENDIX B MATLAB CODE FOR EXAMPLE TWO

```
clear;
clg;

%                          unifineel1
%
% This program will calculate the eigenvalues, eigenvectors
% natural frequencies, and frequency response function matrix
% for a three degree of freedom at each node element.
% The system is modeled with beam elements that are
% aligned in the same plane but at any angle (2-D)
%          •--------•
%          |        |
%          |        |
%          •--------•
%          |        |
%          |        |
%          •--------•
% This program works for a beam element modeled with six general
% coordinates and thus six DOF.
%          (|-•-------•-|)
%
% The user must enter the following data to meet the beam configuration.
% (E) youngs modulus psi
% (I) area moment of inertia in^4
% (WTD) weight density lbf/in^3
% (A) cross sectional area in^2
% conductivity [ the node connection mapping ]
% node coordinates [ cartesian coordinates for each node ]
% (bb) proportional damping constant
% call the data file
minihull_data2A
%
% Start the program clock and flops to determine program running
% time and floating point calculations.
t0=clock;
flops(0);
%
% calculate the number of beam elements
a=size(con);
numel=a(1);
%
% calculate the number of beam elements porportionally damped
aa=size(dcon);
numel_damp=aa(1);
```

```
%
% calculate the number of nodes
b=size(coord);
nodes=b(1);
%
% convert the coordinates in to the correct units (in.)
coord=coord*12;
%
% calculate the beam element lengths and beam angles in radians
for i=1:numel
IC=con(i,1);
ID=con(i,2);
l(1,i)=sqrt((coord(ID,1)-coord(IC,1))^2+(coord(ID,2)-coord(IC,2))^2);
DX(i)=coord(ID,1)-coord(IC,1);
DY(i)=coord(ID,2)-coord(IC,2);
  if DX(i)>=0 & DY(i)>=0;
     t(1,i)=acos(DX(i)/l(1,i));
  elseif DX(i)<0 & DY(i)>=0;
     t(1,i)=acos(DY(i)/l(1,i))+pi/2;
  elseif DX(i)<0 & DY(i)<=0;
     t(1,i)=acos(abs(DX(i))/l(1,i))+pi;
  else
     t(1,i)=acos(abs(DY(i))/l(1,i))+(3*pi/2);
  end;
end;
%
% call trig function
[c,s]=ftrig(t,numel);
%
% calculate radius of gyration
for i=1:numel
r(1,i)=sqrt(I(i)/A(i));
end;
%
% create the global matrix which is all zeroes
%
kg=[zeros(nodes*3,nodes*3)];
mg=[zeros(nodes*3,nodes*3)];
%
% assembel the elemental matricies to the global matrix
%
for i=1:numel
[kel,mel]=felement6(l(i),WTD(i),I(i),E(i),A(i),r(i),c(i),s(i));
%
v=con(i,1);
w=con(i,2);
%
kg(3*v-2:3*v,3*v-2:3*v) = kg(3*v-2:3*v,3*v-2:3*v) + kel(1:3,1:3);
kg(3*v-2:3*v,3*w-2:3*w) = kg(3*v-2:3*v,3*w-2:3*w) + kel(1:3,4:6);
kg(3*w-2:3*w,3*v-2:3*v) = kg(3*w-2:3*w,3*v-2:3*v) + kel(4:6,1:3);
kg(3*w-2:3*w,3*w-2:3*w) = kg(3*w-2:3*w,3*w-2:3*w) + kel(4:6,4:6);
```

```
%
mg(3*v-2:3*v,3*v-2:3*v) = mg(3*v-2:3*v,3*v-2:3*v) + mel(1:3,1:3);
mg(3*v-2:3*v,3*w-2:3*w) = mg(3*v-2:3*v,3*w-2:3*w) + mel(1:3,4:6);
mg(3*w-2:3*w,3*v-2:3*v) = mg(3*w-2:3*w,3*v-2:3*v) + mel(4:6,1:3);
mg(3*w-2:3*w,3*w-2:3*w) = mg(3*w-2:3*w,3*w-2:3*w) + mel(4:6,4:6);
end
%
%
% apply structural prop. damping to the k matrix and set global
% k matrix to equal damped matrix
%
%
% calculate the beam element lengths and beam angles in radians
% for the damped beams
for i=1:numel_damp
IC=dcon(i,1);
ID=dcon(i,2);
l(1,i)=sqrt((coord(ID,1)-coord(IC,1))^2+(coord(ID,2)-coord(IC,2))^2);
DX(i)=coord(ID,1)-coord(IC,1);
DY(i)=coord(ID,2)-coord(IC,2);
  if DX(i)>=0 & DY(i)>=0;
     t(1,i)=acos(DX(i)/l(1,i));
  elseif DX(i)<0 & DY(i)>=0;
     t(1,i)=acos(DY(i)/l(1,i))+pi/2;
  elseif DX(i)<0 & DY(i)<=0;
     t(1,i)=acos(abs(DX(i))/l(1,i))+pi;
  else
     t(1,i)=acos(abs(DY(i))/l(1,i))+(3*pi/2);
  end;
end;
%
% call trig function
[c,s]=ftrig(t,numel_damp);
%
% calculate radius of gyration
for i=1:numel_damp
r(1,i)=sqrt(I(i)/A(i));
end;
%
%
%
kgd=kg;
for u=1:numel_damp
[kel]=felement6(l(u),WTD(u),I(u),E(u),A(u),r(u),c(u),s(u));
%
v=dcon(u,1);
w=dcon(u,2);
%
kgd(3*v-2:3*v,3*v-2:3*v) = kgd(3*v-2:3*v,3*v-2:3*v) + j*bb*kel(1:3,1:3);
kgd(3*v-2:3*v,3*w-2:3*w) = kgd(3*v-2:3*v,3*w-2:3*w) + j*bb*kel(1:3,4:6);
kgd(3*w-2:3*w,3*v-2:3*v) = kgd(3*w-2:3*w,3*v-2:3*v) + j*bb*kel(4:6,1:3);
```

```
kgd(3*w-2:3*w,3*w-2:3*w) = kgd(3*w-2:3*w,3*w-2:3*w) + j*bb*kel(4:6,4:6) ;
%
end
%
% apply the boundary conditions
% the user must adjust the global matrix to meet the boundary conditions
%
% to delete rows
kg([BC],:) = [];
kgd([BC],:) = [];
mg([BC],:) = [];
% to delete columns
kg(:,[BC]) = [];
kgd(:,[BC]) = [];
mg(:,[BC]) = [];
%
% call the function and calculate eigenvectors and the eigenvalues
% which are the mode shapes and undamped natural frequency in
%(rad/sec)^2
%
[lambda,phi]=fgmodes(kg,mg);
% convert the eigenvalues to nat frequency in (rad/sec) and
% and hertz(1/sec)
%
omega = sqrt(lambda);
freq = omega/(2*pi);
%
% construct the frequency response plot over the frequencies
% of interest
count=0;
for Comega=.1:.1:22
count=count+1;
Z=kgd-Comega^2*mg;
H=inv(Z);
%
% determines the coordinate of intrest to plot
HH(count)=H(8,8);
end;
% end the program clock and flops
etime(clock,t0),flops
Comega=.1:.1:22;
Freq=Comega/(2*pi);
plot(Freq,20*log(HH)),grid
xlabel(' Frequency Hz ')
ylabel('FRF at coordinate of interest dB ')
%
% END


%              minihull_data2A
%
```

```
%
% This is the data for the finite element program with three
% degrees of freedom at a node.
%
%       (|-•---------•-|)
%
% The data will be in the form of
% - (E) youngs modulus psi
% - (I) area moment of inertia in^4
% - (WTD) weight density lbf/in^3
% - (A) cross sectional area in^2
% - conductivity [ the node connection mapping ]
% - node coordinates [ cartesian coords for each node in ft. ]
%                      the main program will convert to in..
% - (bb) structural proportional damping constant
%
E=[30 30 30 30 30 30 30 30 30 30 30 30 30 30 30 30 30]*1e6;
I=[.02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083
.02083 .02083 .02083 .02083 .02083 .02083 .02083];%1/12bh^3 b=2,
h=.5
A=[1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1];
WTD=[.2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832
.2832 .2832 .2832 .2832 .2832 .2832];
bb=0.01;
con=[1,2;
     2,3;
     3,4;
     4,5;
     5,6;
     6,7;
     7,8;
     8,9;
     9,10;
     10,11;
     11,12;
     12,1;
     11,13;
     13,14;
     14,15;
     15,5;
     13,12;
     15,4];
% this determines what elements have damping
dcon=[1,2;
      2,3;
      3,4;
      4,5;
      5,6;
      6,7;
      7,8;
      8,9;
```

```
      9,10;
      10,11;
      11,12;
      12,1;
      11,13;
      13,14;
      14,15;
      15,5;
      13,12;
      15,4];
%
coord=[4,0;
       8,0;
       12,0;
       16,4;
       16,8;
       16,12;
       12,16;
       8,16;
       4,16;
       0,12;
       0,8;
       0,4;
       4,8;
       8,8;
       12,8];
%
BC=[ ];


%              inner_struc2A
%
%
% This is the data for the finite element program with three
% degrees of freedom at a node.
%
%       (|-•---------•-|)
%
% The data will be in the form of
% - (E) youngs modulus psi
% - (I) area moment of inertia in^4
% - (WTD) weight density lbf/in^3
% - (A) cross sectional area in^2
% - conductivity [ the node connection mapping ]
% - node coordinates [ cartesian coords for each node in ft. ]
%                      the main program will convert to in..
% - (bb) structural proportional damping constant
E=[30 30 30 30 30 30]*1e6;
I=[ .02083 .02083 .02083 .02083 .02083 .02083];%1/12bh^3 b=2, h=.5
A=[ 1 1 1 1 1 1];
WTD=[ .2832 .2832 .2832 .2832 .2832 .2832];
```

```
bb=0.01;
con=[1,2;
     2,3;
     3,4;
     4,5;
     2,6;
     4,7];
%
% this determines what elements have damping
dcon=[1,2;
      2,3;
      3,4;
      4,5;
      2,6;
      4,7];
%
coord=[0,4;
       4,4;
       8,4;
       12,4;
       16,4;
       0,0;
       16,0];
%
BC=[ ];


%              out_struc2A
%
%
% This is the data for the finite element program with three
% degrees of freedom at a node.
%
%       (|-•---------•-|)
%
% the data will be in the form of
% - (E) youngs modulus psi
% - (I) area moment of inertia in^4
% - (WTD) weight density lbf/in^3
% - (A) cross sectional area in^2
% - conductivity [ the node connection mapping ]
% - node coordinates [ cartesian coords for each node in ft. ]
%                      the main program will convert to in..
% - (bb) structural proportional damping constant
E=[30 30 30 30 30 30 30 30 30 30 30 30]*1e6;
I=[ .02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083
.02083 .02083 .02083];%1/12bh^3 b=2, h=.5
A=[ 1 1 1 1 1 1 1 1 1 1 1 1];
WTD=[.2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832
.2832];
bb=0.01;
```

```
%
con=[1,2;
     2,3;
     3,4;
     4,5;
     5,6;
     6,7;
     7,8;
     8,9;
     9,10;
     10,11;
     11,12;
     12,1];
% this determines what elements have damping
dcon=[1,2;
      2,3;
      3,4;
      4,5;
      5,6;
      6,7;
      8,9;
      9,10;
      10,11;
      11,12;
      12,1];
%
coord=[4,0;
       8,0;
       12,0;
       16,4;
       16,8;
       16,12;
       12,16;
       8,16;
       4,16;
       0,12;
       0,8;
       0,4];
%
BC=[ ];


clear
clg
%                    FRF_Synth2A
%
% This is example 2, dynamic direct coupling using the boolean
% mapping matrix to synthesize two structures together.
%
% Load data from running unifineel program for each substructure
% where the K and M matrix for each structure is saved
```

```
load ex2Aa.mat     % k1 m1 is stored here
load ex2Ab.mat     % k2 m2 is stored here
%
%  We need to create a single FRF matrix representing
%  both substructures in the form:
%
%       [hee] =  [ h(i,i)   h(i,c) ]
%                [ h(c,i)   h(c,c) ]
%
%
%  So we create arrays containing the DOF numbers of our original
%  models which correspond the the "c" and "i" coordinates for
%  each substructure.
%
% Call the synthesis data file which contains the
% internal coordinates and connection coordinates for each substructure.
%
% i1= internal coords of sub structure 1
% i2= internal coords of sub structure 2
% c1= connection coords of sub structure 1
% c2= connection coords of sub structure 2
%
FRF_Synth_data2A
%
t0=clock;
flops(0);
count=0;
for Comega=.1:.3:60
count=count+1;
%
%            Form Frequency Response Models for Each Substructure
%            ~~~~ ~~~~~~~~~ ~~~~~~~~ ~~~~~~ ~~~ ~~~~ ~~~~~~~~~~~~
z1=k1-Comega^2*m1;
z2=k2-Comega^2*m2;
h1=inv(z1);
h2=inv(z2);
%
a=size(i1);
b=size(c1);
c=size(i2);
d=size(c2);
%
aa=a(2);
bb=b(2);
cc=c(2);
dd=d(2);
%
%   Remember, we are trying to calculate the following:
%
%   hee* = hee - hec * M * inv( hccr ) * M' * hce
%   hccr = M' * hcc * M
```

```
%
%  So we need to assemble [hee], [hec], [hce] and [hcc] using the
%  the coordinate sets we just defined.
%  These matrices contain the FRF data for both substructures
%  prior to coupling, i.e the pre-synthesis FRF data.
%
%
%                    Coordinate Partitioning
%                    ---------- ------------
%  Build up uncoupled FRF matrix and sub-partitions:
%
hee = [h1(i1,i1)  zeros(aa,cc)   h1(i1,c1)  zeros(aa,dd);
       zeros(cc,aa)   h2(i2,i2)  zeros(cc,bb)  h2(i2,c2);
       h1(c1,i1)   zeros(bb,cc)   h1(c1,c1)  zeros(bb,dd);
       zeros(dd,aa)  h2(c2,i2)  zeros(dd,bb)  h2(c2,c2)];
%
%
hec = [h1(i1,c1)  zeros(aa,dd);
       zeros(cc,bb)  h2(i2,c2);
       h1(c1,c1)   zeros(bb,dd);
       zeros(dd,bb)  h2(c2,c2)];
%
%
hcc =  [h1(c1,c1) zeros(bb,dd);
        zeros(dd,bb)  h2(c2,c2)];
%
hce =  [h1(c1,i1)   zeros(bb,cc)   h1(c1,c1)  zeros(bb,dd);
       zeros(dd,aa)  h2(c2,i2)  zeros(dd,bb)  h2(c2,c2)];
%
%
%
%  We can now perform the synthesis:
hccr=M' * hcc * M;
heestar = hee - hec * M * inv( hccr ) * M' * hce;
%
% remove the redundent information
heestar = heestar(1:nodes*3,1:nodes*3);
% look at the coordinate of interest
HH(count)=heestar(8,8);
end;
etime(clock,t0),flops
Comega=.1:.3:60;
Freq=Comega/(2*pi);
plot(Freq,20*log(HH)),grid
xlabel(' Frequency Hz ')
ylabel('FRF at coordinate of interest dB')


%                    FRF_Synth_data2A
% This is the data file for the synthesis program.
%
```

```
% The following data will be provided by this file.
%
% i1- internal coords sub structure 1
% i2- internal coords sub structure 2
% c1- connection coords sub structure 1
% c2- connection coords sub structure 2
%
i1=[1 2 3 4 5 6 7 8 9 16 17 18 19 20 21 22 23 24 25 26 27 28 29 30];
c1=[10 11 12 13 14 15 31 32 33 34 35 36];
i2=[4 5 6 7 8 9 10 11 12];
c2=[1 2 3 13 14 15 16 17 18 19 20 21];
%
% The following is the mapping matrix.
% The mapping matrix is not general and is
% case specific.
%
M=[eye(12);
   0 0 0 0 0 0 -1 0 0 0 0 0;
   0 0 0 0 0 0 0 -1 0 0 0 0;
   0 0 0 0 0 0 0 0 -1 0 0 0;
   0 0 0 -1 0 0 0 0 0 0 0 0;
   0 0 0 0 -1 0 0 0 0 0 0 0;
   0 0 0 0 0 -1 0 0 0 0 0 0;
   0 0 0 0 0 0 0 0 0 -1 0 0;
   0 0 0 0 0 0 0 0 0 0 -1 0;
   0 0 0 0 0 0 0 0 0 0 0 -1;
   -1 0 0 0 0 0 0 0 0 0 0 0;
   0 -1 0 0 0 0 0 0 0 0 0 0;
   0 0 -1 0 0 0 0 0 0 0 0 0];
%
%
% Enter the number of unrestrained nodes of the synthesized
% structure.
nodes = 15;
```

APPENDIX C MATLAB CODE FOR EXAMPLE THREE

```
clear
clg
%
%                          unifineel
%
% This program will calculate the eigenvalues,eigenvectors
% natural frequencies, and frequency response function matrix
% for a three degree of freedom at each node element.
% The system is modeled with beam elements that are
% aligned in the same plane but at any angle (2-D).
%          •--------•
%          |        |
%          |        |
%          •--------•
%          |        |
%          |        |
%          •--------•
% This program works for a beam element modeled with six general
% coordinates and thus six DOF.
%          (|-•--------•-|)
%
% the user must enter the following data to meet the beam configuration
% (E) youngs modulus psi
% (I) area moment of inertia in^4
% (WTD) weight density lbf/in^3
% (A) cross sectional area in^2
% conductivity [ the node connection mapping ]
% node coordinates [ cartesian coordinates for each node ]
% (bb) proportional damping constant
clear;
% call the data file
minihull_data3
%
% start the program clock and flops to determine program running
% time and floating point calculations
t0=clock;
flops(0);
%
% calculate the number of beam elements
a=size(con);
numel=a(1);
%
% calculate the number of beam elements porportionally damped
aa=size(dcon);
```

```
numel_damp=aa(1);
%
% calculate the number of nodes.
b=size(coord);
nodes=b(1);
%
% convert the coordinates in to the correct units (in.)
coord=coord*12;

% calculate the beam element lengths and beam angles in radians
for i=1:numel
IC=con(i,1);
ID=con(i,2);
l(1,i)=sqrt((coord(ID,1)-coord(IC,1))^2+(coord(ID,2)-coord(IC,2))^2);
DX(i)=coord(ID,1)-coord(IC,1);
DY(i)=coord(ID,2)-coord(IC,2);
  if DX(i)>=0 & DY(i)>=0;
     t(1,i)=acos(DX(i)/l(1,i));
  elseif DX(i)<0 & DY(i)>=0;
     t(1,i)=acos(DY(i)/l(1,i))+pi/2;
  elseif DX(i)<0 & DY(i)<=0;
     t(1,i)=acos(abs(DX(i))/l(1,i))+pi;
  else
     t(1,i)=acos(abs(DY(i))/l(1,i))+(3*pi/2);
  end;
end;
%
% call trig function
[c,s]=ftrig(t,numel);
%
% calculate radius of gyration
for i=1:numel
r(1,i)=sqrt(I(i)/A(i));
end;
%
% create the global matrix which is all zeroes.
%
kg=[zeros(nodes*3,nodes*3)];
mg=[zeros(nodes*3,nodes*3)];
%
% assembel the elemental matricies to the global matrix.
%
for i=1:numel
[kel,mel]=felement6(l(i),WTD(i),I(i),E(i),A(i),r(i),c(i),s(i));
%
v=con(i,1);
w=con(i,2);
%
kg(3*v-2:3*v,3*v-2:3*v) = kg(3*v-2:3*v,3*v-2:3*v) + kel(1:3,1:3);
kg(3*v-2:3*v,3*w-2:3*w) = kg(3*v-2:3*v,3*w-2:3*w) + kel(1:3,4:6);
kg(3*w-2:3*w,3*v-2:3*v) = kg(3*w-2:3*w,3*v-2:3*v) + kel(4:6,1:3);
```

```
kg(3*w-2:3*w,3*w-2:3*w) = kg(3*w-2:3*w,3*w-2:3*w) + kel(4:6,4:6);
%
mg(3*v-2:3*v,3*v-2:3*v) = mg(3*v-2:3*v,3*v-2:3*v) + mel(1:3,1:3);
mg(3*v-2:3*v,3*w-2:3*w) = mg(3*v-2:3*v,3*w-2:3*w) + mel(1:3,4:6);
mg(3*w-2:3*w,3*v-2:3*v) = mg(3*w-2:3*w,3*v-2:3*v) + mel(4:6,1:3);
mg(3*w-2:3*w,3*w-2:3*w) = mg(3*w-2:3*w,3*w-2:3*w) + mel(4:6,4:6);
end
%
%
% apply structural prop. damping to the k matrix and set global
% k matrix to equal damped matrix
%
%
%  calculate the beam element lengths and beam angles in radians
% for the damped beams
for i=1:numel_damp
IC=dcon(i,1);
ID=dcon(i,2);
l(1,i)=sqrt((coord(ID,1)-coord(IC,1))^2+(coord(ID,2)-coord(IC,2))^2);
DX(i)=coord(ID,1)-coord(IC,1);
DY(i)=coord(ID,2)-coord(IC,2);
  if DX(i)>=0 & DY(i)>=0;
     t(1,i)=acos(DX(i)/l(1,i));
  elseif DX(i)<0 & DY(i)>=0;
     t(1,i)=acos(DY(i)/l(1,i))+pi/2;
  elseif DX(i)<0 & DY(i)<=0;
     t(1,i)=acos(abs(DX(i))/l(1,i))+pi;
  else
     t(1,i)=acos(abs(DY(i))/l(1,i))+(3*pi/2);
  end;
end;
%
% call trig function
[c,s]=ftrig(t,numel_damp);
%
% calculate radius of gyration
for i=1:numel_damp
r(1,i)=sqrt(I(i)/A(i));
end;
%
%
%
kgd=kg;
for u=1:numel_damp
[kel]=felement6(l(u),WTD(u),I(u),E(u),A(u),r(u),c(u),s(u));
%
v=dcon(u,1);
w=dcon(u,2);
%
kgd(3*v-2:3*v,3*v-2:3*v) = kgd(3*v-2:3*v,3*v-2:3*v) + j*bb*kel(1:3,1:3);
kgd(3*v-2:3*v,3*w-2:3*w) = kgd(3*v-2:3*v,3*w-2:3*w) + j*bb*kel(1:3,4:6);
```

```
kgd(3*w-2:3*w,3*v-2:3*v) = kgd(3*w-2:3*w,3*v-2:3*v) + j*bb*kel(4:6,1:3) ;
kgd(3*w-2:3*w,3*w-2:3*w) = kgd(3*w-2:3*w,3*w-2:3*w) + j*bb*kel(4:6,4:6) ;
%
end
%
% apply the initial conditions
% the user must adjust the global matrix to meet the initial conditions
%
% to delete rows
kg([BC],:) = [ ];
kgd([BC],:) = [ ];
mg([BC],:) = [ ];
% to delete columns
kg(:,[BC]) = [ ];
kgd(:,[BC]) = [ ];
mg(:,[BC]) = [ ];
%
% call the function and calculate eigenvectors and the eigenvalues
% which are the mode shapes and undamped natural frequency in
(rad/sec)^2
%
[lambda,phi]=fgmodes(kg,mg);
% this now converts the eigenvalues to nat frequency in (rad/sec) and
% hertz(1/sec)
%
omega = sqrt(lambda);
freq = omega/(2*pi);
%
% constuct the frequency response plot over the frequencies
% of interest
count=0;
for Comega=.1:.3:40
count=count+1;
Z=kgd-Comega^2*mg;
H=inv(Z);
%
% this determines the coordinate of intrest to plot
HH(count)=H(14,14);
end;
% end the program clock and flops
etime(clock,t0),flops
Comega=.1:.3:40;
Freq=Comega/(2*pi);
plot(Freq,20*log(HH)),grid
xlabel(' Frequency Hz ')
ylabel('FRF at coordinate of interest dB')
% END


%              minihull_data3
%
```

```
% This is the data file for example 3
%
% The data will be in the form of
% - (E) youngs modulus psi
% - (I) area moment of inertia in^4
% - (WTD) weight density lbf/in^3
% - (A) cross sectional area in^2
% - conductivity [ the node connection mapping ]
% - node coordinates [ cartesian coords for each node in ft. ]
%                      the main program will convert to in..
% - (bb) structural proportional damping constant
%
E=[ 30 30 30 30 30 30 30 30 30 30 30 30 30 30 30 30 30]*1e6;
I=[ .02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083
.02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083];%1/12bh^3 b=2,
h=.5
A=[ 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1];
WTD=[ .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832
.2832 .2832 .2832 .2832 .2832 .2832];
bb=0.01;
con=[1,2;
     2,3;
     3,4;
     4,5;
     5,6;
     6,7;
     7,8;
     8,9;
     9,10;
     10,11;
     11,12;
     12,1;
     11,13;
     13,14;
     14,15;
     15,5;
     13,12];
% this deteremines what elements have damping
dcon=[1,2;
      2,3;
      3,4;
      4,5;
      5,6;
      6,7;
      7,8;
      8,9;
      9,10;
      10,11;
      11,12;
      12,1;
      11,13;
```

```
       13,14;
       14,15;
       15,5;
       15,12];
%
coord=[4,0;
       8,0;
       12,0;
       16,4;
       16,8;
       16,12;
       12,16;
       8,16;
       4,16;
       0,12;
       0,8;
       0,4;
       4,8;
       8,8;
       12,8];
%
BC=[];


%              hull_mod3
%
%
% This is the data file for the beam modification.
%
% The data will be in the form of
% - (E) youngs modulus psi
% - (I) area moment of inertia in^4
% - (WTD) weight density lbf/in^3
% - (A) cross sectional area in^2
% - conductivity [ the node connection mapping ]
% - node coordinates [ cartesian coords for each node in ft. ]
%                      the main program will convert to in..
% - (bb) structural proportional damping constant
%
E=[30 ]*1e6;
I=[.02083 ];
A=[1 ];
WTD=[.2832];
bb=0.01;
con=[1,2];
% this determines what elements have damping
dcon=[1,2];
%
coord=[4,0;
       0,4];
%
```

```
BC=[];


%              red_mod3
%
%
% This is the data file for the main substructure.
%
% The data will be in the form of
% - (E) youngs modulus psi
% - (I) area moment of inertia in^4
% - (WTD) weight density lbf/in^3
% - (A) cross sectional area in^2
% - conductivity [ the node connection mapping ]
% - node coordinates [ cartesian coords for each node in ft. ]
%                      the main program will convert to in..
% - (bb) structural proportional damping constant
%
E=[30 30 30 30 30 30 30 30 30 30 30 30 30 30 30 30 30]*1e6;
I=[.02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083
.02083 .02083 .02083 .02083 .02083 .02083 .02083];%1/12bh^3 b=2,
h=.5
A=[1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1];
WTD=[.2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832
.2832 .2832 .2832 .2832 .2832 .2832];
bb=0.01;
%
con=[1,2;
     2,3;
     3,4;
     4,5;
     5,6;
     6,7;
     7,8;
     8,9;
     9,10;
     10,11;
     11,12;
     12,1;
     11,13;
     13,14;
     14,15;
     15,5;
     13,12;
     15,4];
% this determines what elements have damping
dcon=[1,2;
      2,3;
      3,4;
      4,5;
      5,6;
```

```
      6,7;
      7,8;
      8,9;
      9,10;
      10,11;
      11,12;
      12,1;
      11,13;
      13,14;
      14,15;
      15,5;
      13,12;
      15,4];
%
coord=[4,0;
       8,0;
       12,0;
       16,4;
       16,8;
       16,12;
       12,16;
       8,16;
       4,16;
       0,12;
       0,8;
       0,4;
       4,8;
       8,8;
       12,8];
%
BC=[];


% This program will modify a substructure by removing a beam
% element from the structure.
%
%                    program FRF_MOD3
clear;
clg;
load ex3a.mat    % k1, m1 are stored here
load ex3b.mat    % k2, m2 are stored here
% call the data file
MOD_DAT3
%
% we need to partion the H matrix of the structure to be
% modified in the following way
%
%     [ ii | ic ]
%     -----------
%     [ ci | cc ]
%
```

```
%      [hee] = [ ii ic
%                ci cc ]
%
%      [hec] = [ ic
%                cc ]
%
%      [hcc] = [ cc ]
%
%
t0=clock;
flops(0);
count=0;
for Comega=.1:.3:40
count=count+1;
z1=k1-Comega^2*m1;
z=k2-Comega^2*m2;
h=inv(z1);
%
hee = [h(ic,ic)  h(ic,cc);
       h(cc,ic)  h(cc,cc)];
%
hec = [h(ic,cc);
       h(cc,cc)];
%
hcc = [h(cc,cc)];
%
hce = [h(cc,ic) h(cc,cc)];
%
% this is for adding a component
%heestar = hee - hec * inv(inv(z) + hcc) * hce;
%
% this is for removing a component
heestar = hee - hec * inv(hcc - inv(z)) * hce;
%
HH(count)=heestar(11,11);
end;
etime(clock,t0),flops
Comega=.1:.3:40;
Freq=Comega/(2*pi);
plot(Freq,20*log(HH)),grid
xlabel(' Frequency Hz ')
ylabel('FRF at coordinate  of interest dB ')


%                          MOD_DAT3

% This is the data file for the modification program,
% the following data will be provided by this file
%
% ic- internal coords of synthesized structure
% cc- connection coords of synthesized structure
```

```
%
ic=[1 2 3 4 5 6 7 8 9 13 14 15 16 17 18 19 20 21 22 23 24 25 26 27 28 29
30 31 32 33 34 35 36 37 38 39 40 41 42 ];
cc=[10 11 12 43 44 45];
```

APPENDIX D MATLAB CODE FOR EXAMPLE FOUR

```
clear
clg

%                          unifineel1
%
% This program will calculate the eigenvalues, eigenvectors
% natural frequencies, and frequency response function matrix
% for a three degree of freedom at each node element.
% The system is modeled with beam elements that are
% aligned in the same plane but at any angle (2-D).
%          •--------•
%          |        |
%          |        |
%          •--------•
%          |        |
%          |        |
%          •--------•
% This program works for a beam element modeled with six general
% coordinates and thus six DOF.
%          (|-•--------•-|)
%
% the user must enter the following data to meet the beam configuration
% (E) youngs modulus psi
% (I) area moment of inertia in^4
% (WTD) weight density lbf/in^3
% (A) cross sectional area in^2
% conductivity [ the node connection mapping ]
% node coordinates [ cartesian coordinates for each node ]
% (bb) proportional damping constant
clear;
% call the data file
minihull_data4
%
% start the program clock and flops to determine program running
% time and floating point calculations
t0=clock;
flops(0);
%
% calculate the number of beam elements
a=size(con);
numel=a(1);
%
% calculate the number of beam elements porportionally damped
aa=size(dcon);
```

```
numel_damp=aa(1);
%
% calculate the number of nodes.
b=size(coord);
nodes=b(1);
%
% convert the coordinates in to the correct units (in.)
coord=coord*12;
%
% now calculate the beam element lengths and beam angles in radians
for i=1:numel
IC=con(i,1);
ID=con(i,2);
l(1,i)=sqrt((coord(ID,1)-coord(IC,1))^2+(coord(ID,2)-coord(IC,2))^2);
DX(i)=coord(ID,1)-coord(IC,1);
DY(i)=coord(ID,2)-coord(IC,2);
  if DX(i)>=0 & DY(i)>=0;
     t(1,i)=acos(DX(i)/l(1,i));
  elseif DX(i)<0 & DY(i)>=0;
     t(1,i)=acos(DY(i)/l(1,i))+pi/2;
  elseif DX(i)<0 & DY(i)<=0;
     t(1,i)=acos(abs(DX(i))/l(1,i))+pi;
  else
     t(1,i)=acos(abs(DY(i))/l(1,i))+(3*pi/2);
  end;
end;
%
% call trig function
[c,s]=ftrig(t,numel);
%
% calculate radius of gyration
for i=1:numel
r(1,i)=sqrt(I(i)/A(i));
end;
%
% create the global matrix which is all zeroes.
%
kg=[zeros(nodes*3,nodes*3)];
mg=[zeros(nodes*3,nodes*3)];
%
% assembel the elemental matricies to the global matrix.
%
for i=1:numel
[kel,mel]=felement6(l(i),WTD(i),I(i),E(i),A(i),r(i),c(i),s(i));
%
v=con(i,1);
w=con(i,2);
%
kg(3*v-2:3*v,3*v-2:3*v) = kg(3*v-2:3*v,3*v-2:3*v) + kel(1:3,1:3);
kg(3*v-2:3*v,3*w-2:3*w) = kg(3*v-2:3*v,3*w-2:3*w) + kel(1:3,4:6);
kg(3*w-2:3*w,3*v-2:3*v) = kg(3*w-2:3*w,3*v-2:3*v) + kel(4:6,1:3);
```

```
kg(3*w-2:3*w,3*w-2:3*w) = kg(3*w-2:3*w,3*w-2:3*w) + kel(4:6,4:6);
%
mg(3*v-2:3*v,3*v-2:3*v) = mg(3*v-2:3*v,3*v-2:3*v) + mel(1:3,1:3);
mg(3*v-2:3*v,3*w-2:3*w) = mg(3*v-2:3*v,3*w-2:3*w) + mel(1:3,4:6);
mg(3*w-2:3*w,3*v-2:3*v) = mg(3*w-2:3*w,3*v-2:3*v) + mel(4:6,1:3);
mg(3*w-2:3*w,3*w-2:3*w) = mg(3*w-2:3*w,3*w-2:3*w) + mel(4:6,4:6);
end
%
% apply structural prop. damping to the k matrix and set global
% k matrix to equal damped matrix
%
% calculate the beam element lengths and beam angles in radians
% for the damped beams
for i=1:numel_damp
IC=dcon(i,1);
ID=dcon(i,2);
l(1,i)=sqrt((coord(ID,1)-coord(IC,1))^2+(coord(ID,2)-coord(IC,2))^2);
DX(i)=coord(ID,1)-coord(IC,1);
DY(i)=coord(ID,2)-coord(IC,2);
  if DX(i)>=0 & DY(i)>=0;
     t(1,i)=acos(DX(i)/l(1,i));
  elseif DX(i)<0 & DY(i)>=0;
     t(1,i)=acos(DY(i)/l(1,i))+pi/2;
  elseif DX(i)<0 & DY(i)<=0;
     t(1,i)=acos(abs(DX(i))/l(1,i))+pi;
  else
     t(1,i)=acos(abs(DY(i))/l(1,i))+(3*pi/2);
  end;
end;
%
% call trig function
[c,s]=ftrig(t,numel_damp);
%
% calculate radius of gyration
for i=1:numel_damp
r(1,i)=sqrt(I(i)/A(i));
end;
%
kgd=kg;
for u=1:numel_damp
[kel]=felement6(l(u),WTD(u),I(u),E(u),A(u),r(u),c(u),s(u));
%
v=dcon(u,1);
w=dcon(u,2);
%
kgd(3*v-2:3*v,3*v-2:3*v) = kgd(3*v-2:3*v,3*v-2:3*v) + j*bb*kel(1:3,1:3);
kgd(3*v-2:3*v,3*w-2:3*w) = kgd(3*v-2:3*v,3*w-2:3*w) + j*bb*kel(1:3,4:6);
kgd(3*w-2:3*w,3*v-2:3*v) = kgd(3*w-2:3*w,3*v-2:3*v) + j*bb*kel(4:6,1:3);
kgd(3*w-2:3*w,3*w-2:3*w) = kgd(3*w-2:3*w,3*w-2:3*w) + j*bb*kel(4:6,4:6);
%
end
```

```
%
% apply the boundary conditions
% the user must adjust the global matrix to meet the boundary conditions
%
% to delete rows
kg([BC],:) = [];
kgd([BC],:) = [];
mg([BC],:) = [];
% to delete columns
kg(:,[BC]) = [];
kgd(:,[BC]) = [];
mg(:,[BC]) = [];
%
% call the function and calculate eigenvectors and the eigenvalues
% which are the mode shapes and undamped natural frequency in
% (rad/sec)^2
%
[lambda,phi]=fgmodes(kg,mg);
% this now converts the eigenvalues to nat frequency in (rad/sec) and
% hertz(1/sec)
%
omega = sqrt(lambda);
freq = omega/(2*pi);
%
% constuct the frequency response plot over the frequencies
% of interest
count=0;
for Comega=.1:.3:53
count=count+1;
Z=kgd-Comega^2*mg;
H=inv(Z);
%
% this determines the coordinate of intrest to plot
HH(count)=H(8,8);
end;
% end the program clock and flops
etime(clock,t0),flops
Comega=.1:.3:53;
Freq=Comega/(2*pi);
plot(Freq,20*log(HH)),grid
xlabel(' Frequency Hz ')
ylabel('FRF at coordinate of interest dB')
% end


%              minihull_data4
%
%
% This is the data file for the finite element program.
%
% the data will be in the form of
```

```
% (E) youngs modulus psi
% (I) area moment of inertia in^4
% (A) cross sectional area in^2
% conductivity [ the node connection mapping ]
% node coordinates [ cartesian coordinates for each node ]
% (bb) proportional damping constant
%
E=[30 30 30 30 30 30 30 30 30 30 30 30 30 30 30 30 30 30]*1e6;
I=[.02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083
.02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083];%1/12bh^3 b=2,
h=.5
A=[1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1];
WTD=[.2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832
.2832 .2832 .2832 .2832 .2832 .2832 .2832];
bb=0.01;
con=[1,2;
     2,3;
     3,4;
     4,5;
     5,6;
     6,7;
     7,8;
     8,9;
     9,10;
     10,11;
     11,12;
     12,1;
     11,13;
     13,14;
     14,15;
     15,5;
     13,12;
     15,4];
% this determines what elements have damping
dcon=[1,2;
      2,3;
      3,4;
      4,5;
      5,6;
      6,7;
      7,8;
      8,9;
      9,10;
      10,11;
      11,12;
      12,1;
      11,13;
      13,14;
      14,15;
      15,5;
      13,12;
```

```
      15,4];
%
coord=[4,0;
       8,0;
       12,0;
       16,4;
       16,8;
       16,12;
       12,16;
       8,16;
       4,16;
       0,12;
       0,8;
       0,4;
       4,8;
       8,8;
       12,8];
%
BC=[];


%              hull_mod4
%
% This is the data file for the finite element program.
% This is the data for the modification
%
% the data will be in the form of
% (E) youngs modulus psi
% (I) area moment of inertia in^4
% (A) cross sectional area in^2
% conductivity [ the node connection mapping ]
% node coordinates [ cartesian coordinates for each node ]
% (bb) proportional damping constant
E=[30 ]*1e6;
I=[.02083 ];
A=[1 ];
WTD=[.2832];
bb=0.01;
con=[1,2];
% this determines what elements have damping
dcon=[1,2];
%
coord=[4,0;
       0,4];
%
BC=[];


%              red_mod4
%
% This is the data file for the main structure to be modified.
```

```
%
% the data will be in the form of
% (E) youngs modulus psi
% (I) area moment of inertia in^4
% (A) cross sectional area in^2
% conductivity [ the node connection mapping ]
% node coordinates [ cartesian coordinates for each node ]
% (bb) proportional damping constant
%
E=[ 30 30 30 30 30 30 30 30 30 30 30 30 30 30 30 30 30]*1e6;
I=[ .02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083
.02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083];%1/12bh^3 b=2,
h=.5
A=[ 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1];
WTD=[ .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832
.2832 .2832 .2832 .2832 .2832 .2832];
bb=0.01;
%
con=[1,2;
     2,3;
     3,4;
     4,5;
     5,6;
     6,7;
     7,8;
     8,9;
     9,10;
     10,11;
     11,12;
     12,1;
     11,13;
     13,14;
     14,15;
     15,5;
     13,12];
% this determines what elements have damping
dcon=[1,2;
      2,3;
      3,4;
      4,5;
      5,6;
      6,7;
      7,8;
      8,9;
      9,10;
      10,11;
      11,12;
      12,1;
      11,13;
      13,14;
      14,15;
```

```
      15,5;
      13,12];
%
coord=[4,0;
       8,0;
       12,0;
       16,4;
       16,8;
       16,12;
       12,16;
       8,16;
       4,16;
       0,12;
       0,8;
       0,4;
       4,8;
       8,8;
       12,8];
%
BC=[];


clear;
clg;

%                  program FRF_MOD4

% This program will modify a structure by adding a beam element.
%
load ex4a.mat   % k1, m1 are stored here
load ex4b.mat   % k2, m2 are stored here
MOD_DAT4
%
% we need to partion the H matrix of the structure to be
% modified in the following way
%
%     [ ii | ic ]
%     -----------
%     [ ci | cc ]
%
%     [hee] = [ ii ic
%               ci cc ]
%
%     [hec] = [ ic
%               cc ]
%
%     [hcc] = [ cc ]
%
t0=clock;
flops(0);
count=0;
```

```
for Comega=.1:.3:53
count=count+1;
z1=k1-Comega^2*m1;
z=k2-Comega^2*m2;
h=inv(z1);
%
hee = [h(ic,ic)  h(ic,cc);
       h(cc,ic)  h(cc,cc)];

hec = [h(ic,cc);
       h(cc,cc)];

hcc = [h(cc,cc)];

hce = [h(cc,ic) h(cc,cc)];

% this is for adding a component
heestar = hee - hec * inv(inv(z) + hcc) * hce;

% this is for removing a component
%heestar = hee - hec * inv(hcc - inv(z)) * hce;

HH(count)=heestar(8,8);
end;
etime(clock,t0),flops
Comega=.1:.3:53;
Freq=Comega/(2*pi);
plot(Freq,20*log(HH)),grid
xlabel(' Frequency Hz ')
ylabel('FRF at coordinate  of interest dB ')


%                          MOD_DAT4
%
% This is the data file for the synthesis program
% the following data will be provided by this file
%
% ic- internal coords of synthesized structure

% cc- connection coords of synthesized structure
%
ic=[1 2 3 4 5 6 7 8 9 13 14 15 16 17 18 19 20 21 22 23 24 25 26 27 28 29
30 31 32 33 34 35 36 37 38 39 40 41 42 ];
cc=[10 11 12 43 44 45];
%
% enter the number of unrestrained nodes of the synthesized
% structure
nodes = 15;
```

APPENDIX E MATLAB CODE FOR EXAMPLE FIVE

```
clear;
clg;

%                          unifineelsprdam
%
% This program will calculate the eigenvalues, eigenvectors
% natural frequencies, and frequency response function matrix
% for a three degree of freedom at each node element.
% The system is modeled with beam elements that are
% aligned in the same plane but at any angle (2-D).
%          •--------•
%          |        |
%          |        |
%          •--------•
%          |        |
%          |        |
%          •--------•
% This program works for a beam element modeled with six general
% coordinates and thus six DOF.
%          (|-•--------•-|)
%
% the user must enter the following data to meet the beam configuration
% (E) youngs modulus psi
% (I) area moment of inertia in^4
% (WTD) weight density lbf/in^3
% (A) cross sectional area in^2
% conductivity [ the node connection mapping ]
% node coordinates [ cartesian coordinates for each node ]
% spring-damper conductivity
% (bb) proportional damping constant
% (B) viscous proportional damping constant
% (k) spring constant
%
% call the data file
minihullsprdam_da5
%
% start the program clock and flops to determine program running
% time and floating point calculations
t0=clock;
flops(0);
%
% calculate the number of beam elements
a=size(con);
numel=a(1);
```

```
%
% calculate the number of beam elements porportionally damped
aa=size(dcon);
numel_damp=aa(1);
%
% calculate the number of nodes.
b=size(coord);
nodes=b(1);
%
% convert the coordinates in to the correct units (in.)
coord=coord*12;
%
% calculate the beam element lengths and beam angles in radians
for i=1:numel
IC=con(i,1);
ID=con(i,2);
l(1,i)=sqrt((coord(ID,1)-coord(IC,1))^2+(coord(ID,2)-coord(IC,2))^2);
DX(i)=coord(ID,1)-coord(IC,1);
DY(i)=coord(ID,2)-coord(IC,2);
  if DX(i)>=0 & DY(i)>=0;
     t(1,i)=acos(DX(i)/l(1,i));
  elseif DX(i)<0 & DY(i)>=0;
     t(1,i)=acos(DY(i)/l(1,i))+pi/2;
  elseif DX(i)<0 & DY(i)<=0;
     t(1,i)=acos(abs(DX(i))/l(1,i))+pi;
  else
     t(1,i)=acos(abs(DY(i))/l(1,i))+(3*pi/2);
  end;
end;
%
% call trig function
[c,s]=ftrig(t,numel);
%
% calculate radius of gyration
for i=1:numel
r(1,i)=sqrt(I(i)/A(i));
end;
%
% create the global matrix which is all zeroes.
%
kg=[zeros(nodes*3,nodes*3)];
mg=[zeros(nodes*3,nodes*3)];
%
% assemble the elemental matricies to the global matrix.
%
for i=1:numel
[kel,mel]=felement6(l(i),WTD(i),I(i),E(i),A(i),r(i),c(i),s(i));
%
v=con(i,1);
w=con(i,2);
%
```

```
kg(3*v-2:3*v,3*v-2:3*v) = kg(3*v-2:3*v,3*v-2:3*v) + kel(1:3,1:3);
kg(3*v-2:3*v,3*w-2:3*w) = kg(3*v-2:3*v,3*w-2:3*w) + kel(1:3,4:6);
kg(3*w-2:3*w,3*v-2:3*v) = kg(3*w-2:3*w,3*v-2:3*v) + kel(4:6,1:3);
kg(3*w-2:3*w,3*w-2:3*w) = kg(3*w-2:3*w,3*w-2:3*w) + kel(4:6,4:6);
%
mg(3*v-2:3*v,3*v-2:3*v) = mg(3*v-2:3*v,3*v-2:3*v) + mel(1:3,1:3);
mg(3*v-2:3*v,3*w-2:3*w) = mg(3*v-2:3*v,3*w-2:3*w) + mel(1:3,4:6);
mg(3*w-2:3*w,3*v-2:3*v) = mg(3*w-2:3*w,3*v-2:3*v) + mel(4:6,1:3);
mg(3*w-2:3*w,3*w-2:3*w) = mg(3*w-2:3*w,3*w-2:3*w) + mel(4:6,4:6);
end
%
% This section will apply structural prop. damping to the k
%  matrix and set global k matrix to equal damped matrix
%
% calculate the beam element lengths and beam angles in radians
% for the damped beams
for i=1:numel_damp
IC=dcon(i,1);
ID=dcon(i,2);
l(1,i)=sqrt((coord(ID,1)-coord(IC,1))^2+(coord(ID,2)-coord(IC,2))^2);
DX(i)=coord(ID,1)-coord(IC,1);
DY(i)=coord(ID,2)-coord(IC,2);
  if DX(i)>=0 & DY(i)>=0;
     t(1,i)=acos(DX(i)/l(1,i));
  elseif DX(i)<0 & DY(i)>=0;
     t(1,i)=acos(DY(i)/l(1,i))+pi/2;
  elseif DX(i)<0 & DY(i)<=0;
     t(1,i)=acos(abs(DX(i))/l(1,i))+pi;
  else
     t(1,i)=acos(abs(DY(i))/l(1,i))+(3*pi/2);
  end;
end;
%
% call trig function
[c,s]=ftrig(t,numel_damp);
%
% calculate radius of gyration
for i=1:numel_damp
r(1,i)=sqrt(I(i)/A(i));
end;
%
kgd=kg;
for u=1:numel_damp
[kel]=felement6(l(u),WTD(u),I(u),E(u),A(u),r(u),c(u),s(u));
%
v=dcon(u,1);
w=dcon(u,2);
%
kgd(3*v-2:3*v,3*v-2:3*v) = kgd(3*v-2:3*v,3*v-2:3*v) + j*bb*kel(1:3,1:3);
kgd(3*v-2:3*v,3*w-2:3*w) = kgd(3*v-2:3*v,3*w-2:3*w) + j*bb*kel(1:3,4:6);
kgd(3*w-2:3*w,3*v-2:3*v) = kgd(3*w-2:3*w,3*v-2:3*v) + j*bb*kel(4:6,1:3);
```

```
kgd(3*w-2:3*w,3*w-2:3*w) = kgd(3*w-2:3*w,3*w-2:3*w) + j*bb*kel(4:6,4:6 ;
%
end
%
% this section will connect a spring-damper system to the global
% stiffness matrix. the spring-damper system is made up of a set of
three springs and dampers that correspond to the degrees of freedom
% at a node. It attaches to the global stiffness matrix based on the
% spring damper connectivity.
%
kgds=kgd;
d = size(sdcon);
numspg = d(1);
count=0;
for Comega=.1:.2:50
count=count+1;
for j=1:numspg;
[kdsprg]=fsprngdamp(k(j),Comega,B(j));
%
x=sdcon(j,1);
y=sdcon(j,2);
%
kgds(3*x-2:3*x,3*x-2:3*x) = kgd(3*x-2:3*x,3*x-2:3*x) + kdsprg(1:3,1:3);
kgds(3*x-2:3*x,3*y-2:3*y) = kgd(3*x-2:3*x,3*y-2:3*y) + kdsprg(1:3,4:6);
kgds(3*y-2:3*y,3*x-2:3*x) = kgd(3*y-2:3*y,3*x-2:3*x) + kdsprg(4:6,1:3);
kgds(3*y-2:3*y,3*y-2:3*y) = kgd(3*y-2:3*y,3*y-2:3*y) + kdsprg(4:6,4:6);
end
%
% apply the boundary conditions
% the user must adjust the global matrix to meet the boundary conditions
%
% to delete rows
kgds([BC],:) = [];
mg([BC],:) = [];
% to delete columns
kgds(:,[BC]) = [];
mg(:,[BC]) = [];
%
Z=kgds-Comega^2*mg;
H=inv(Z);
%
HH(count)=H(8,8);
end;
%
[lambda,phi]=fgmodes(kgds,mg);
% this now converts the eigenvalues to nat frequency in (rad/sec) and
% hertz(1/sec)
%
omega = sqrt(lambda);
freq = omega/(2*pi);
%
```

```
etime(clock,t0),flops
Comega=.1:.2:50;
%
Freq=Comega/(2*pi);
plot(Freq,20*log(HH)),grid
xlabel(' Frequency Hz ')
ylabel('FRF at coordinate of interest dB ')
% end


%              minihullsprdam_data5
%
%
% This is the data for the finite element program.
%
% the data will be in the form of
% - (E) youngs modulus psi
% - (I) area moment of inertia in^4
% - (A) cross sectional area in^2
% - conductivity [ the node connection mapping ]
% - node coordinates [ cartesian coordinates for each node ]
% - (bb) proportional damping constant
% - (k) spring constant lbs/in
% - (B) proportional viscous damping constant
%
E=[30 30 30 30 30 30 30 30 30 30 30 30 30 30 30 30 30 30]*1e6;
I=[.02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083
.02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083];%1/12bh^3 b=2,
h=.5
A=[1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1];
WTD=[.2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832
.2832 .2832 .2832 .2832 .2832 .2832 .2832];
B=[.01 .01 .01 .01];                % porp damp  2%
k=[25 25 25 25];                    % lbs/in
bb=0.02;                            % structural porp damp 2%
con=[1,2;
     2,3;
     3,4;
     4,5;
     5,6;
     6,7;
     7,8;
     8,9;
     9,10;
     10,11;
     11,12;
     12,1;
     13,14;
     14,15;
     15,16;
     16,17;
```

```
      14,18;
      16,19];
% this determines what elements have damping
dcon=[1,2;
      2,3;
      3,4;
      4,5;
      5,6;
      6,7;
      7,8;
      8,9;
      9,10;
      10,11;
      11,12;
      12,1;
      13,14;
      14,15;
      15,16;
      16,17;
      14,18;
      16,19];
% this is the isolator connectivity
sdcon=[11,13;
       17,5;
       12,18;
       19,4];
%
coord=[4,0;
       8,0;
       12,0;
       16,4;
       16,8;
       16,12;
       12,16;
       8,16;
       4,16;
       0,12;
       0,8;
       0,4;
       0,8;
       4,8;
       8,8;
       12,8;
       16,8;
       0,4;
       16,4];
%
BC=[];


%              out_struc5
```

```
%
% This is the data file for the outer structure to be synthesized.
%
% the data will be in the form of
% - (E) youngs modulus psi
% - (I) area moment of inertia in^4
% - (A) cross sectional area in^2
% - conductivity [ the node connection mapping ]
% - node coordinates [ cartesian coordinates for each node ]
% - (bb) proportional damping constant
%
E=[30 30 30 30 30 30 30 30 30 30 30 30]*1e6;
I=[ .02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083
.02083 .02083 .02083];%1/12bh^3 b=2, h=.5
A=[ 1 1 1 1 1 1 1 1 1 1 1 1];
WTD=[.2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832
.2832];
bb=0.02;

con=[1,2;
     2,3;
     3,4;
     4,5;
     5,6;
     6,7;
     7,8;
     8,9;
     9,10;
     10,11;
     11,12;
     12,1];
% this determines what elements have damping
dcon=[1,2;
     2,3;
     3,4;
     4,5;
     5,6;
     6,7;
     7,8;
     8,9;
     9,10;
     10,11;
     11,12;
     12,1];
%
coord=[4,0;
       8,0;
       12,0;
       16,4;
       16,8;
       16,12;
```

```
       12,16;
       8,16;
       4,16;
       0,12;
       0,8;
       0,4];
%
BC=[];


%              inner_struc5
%
%
% This is the data file for the inner structure to be synthesized
%
% the data will be in the form of
% - (E) youngs modulus psi
% - (I) area moment of inertia in^4
% - (WTD) weight density lbf/in^3
% - (A) cross sectional area in^2
% - conductivity [ the node connection mapping ]
% - node coordinates [ cartesian coords for each node in ft. ]
%                      the main program will convert to in..
% - (bb) structural proportional damping constant
%
E=[30 30 30 30 30 30]*1e6;
I=[ .02083 .02083 .02083 .02083 .02083 .02083];%1/12bh^3 b=2, h=.5
A=[ 1 1 1 1 1 1];
WTD=[ .2832 .2832 .2832 .2832 .2832 .2832];
bb=0.02;
con=[1,2;
     2,3;
     3,4;
     4,5;
     2,6;
     4,7];
% this determines what elements have damping
dcon=[1,2;
      2,3;
      3,4;
      4,5;
      2,6;
      4,7];
%
coord=[0,4;
       4,4;
       8,4;
       12,4;
       16,4;
       0,0;
       16,0];
```

```
%
BC=[];


clear;clc;
clg;
%
%                        FRF_INDSPRDAM_5
%
% Load data from running unifineel program for each substructure.
% The K and M matrix for each structure is saved.
load ex5a.mat     % k1 m1 are stored here
load ex5b.mat     % k2 m2 are stored here
%
%  We need to create a single FRF matrix representing
%  both substructures in the form:
%
%       [hee] =  [ h(i,i)   h(i,c) ]
%                [ h(c,i)   h(c,c) ]
%
%  So we create arrays containing the DOF numbers of our original
%  models which correspond the the "c" and "i" coordinates for
%  each substructure.
%
% call the synthesis data file in now which contains the
% internal coordinates and connection coordinates for each sub
% structure.
%
% i1= internal coords of sub structure 1
% i2= internal coords of sub structure 2
% c1= connection coords of sub structure 1
% c2= connection coords of sub structure 2
%
FRF_Indsprdam_data5
%
t0 = clock;
flops(0);
count=0;
for Comega=.1:.2:50
count=count+1;
%
%           Form Frequency Response Models for Each Substructure
%           ---- --------- -------- ------ --- ---- ------------
%
z1=k1-Comega^2*m1;
z2=k2-Comega^2*m2;
h1=inv(z1);
h2=inv(z2);
%
a=size(i1);
b=size(c1);
```

```
c=size(i2);
d=size(c2);
%
aa=a(2);
bb=b(2);
cc=c(2);
dd=d(2);
%
%   Remember, we are trying to calculate the following:
%
%   hee* = hee - hec * M * inv(zr + hccr ) * M' * hce
%   hccr = M' * hcc * M
%   zr = pinv(M) * z * pinv(M') which is just identity matrix size
%                          3 times the number of spring-damp systems
%
%  So we need to assemble [hee], [hec], [hce] and [hcc] using the
%  the coordinate sets we just defined.
%  These matrices contain the FRF data for both substructures
%  prior to coupling, i.e the pre-synthesis FRF data.
%
%                    Coordinate Partitioning
%                    ---------- -------------
%
%  Build up uncoupled FRF matrix and sub-partitions:
%
hee = [h1(i1,i1)  zeros(aa,cc)   h1(i1,c1)  zeros(aa,dd);
      zeros(cc,aa)   h2(i2,i2)  zeros(cc,bb)  h2(i2,c2);
       h1(c1,i1)   zeros(bb,cc)   h1(c1,c1)  zeros(bb,dd);
       zeros(dd,aa)  h2(c2,i2)  zeros(dd,bb)  h2(c2,c2)];
%
hec = [h1(i1,c1)  zeros(aa,dd);
       zeros(cc,bb)  h2(i2,c2);
       h1(c1,c1)   zeros(bb,dd);
       zeros(dd,bb)  h2(c2,c2)];
%
hcc =  [h1(c1,c1) zeros(bb,dd);
        zeros(dd,bb)  h2(c2,c2)];
%
hce=   [h1(c1,i1)   zeros(bb,cc)   h1(c1,c1)  zeros(bb,dd);
       zeros(dd,aa)  h2(c2,i2)  zeros(dd,bb)  h2(c2,c2)];
%
%  We can now perform the synthesis:
zr = (k + j*Comega*B*k) * eye(12);
hccr = M' * hcc * M;
heestar = hee - hec * M * inv(inv(zr) + hccr ) * M' * hce;
HH(count)=heestar(8,8);
end;
etime(clock,t0),flops
%
Comega=.1:.2:50;
Freq=Comega/(2*pi);
```

```
plot(Freq,20*log(HH)),grid
xlabel(' Frequency Hz ')
ylabel('FRF at coordinate of interest dB ')

%                    FRF_Indsprdam_data5
%
% This is the data file for the synthesis program.
% The following data will be provided by this file
%
% i1- internal coords sub structure 1
% i2- internal coords sub structure 2
% c1- connection coords sub structure 1
% c2- connection coords sub structure 2
%
i1=[1 2 3 4 5 6 7 8 9 16 17 18 19 20 21 22 23 24 25 26 27 28 29 30];
c1=[10 11 12 13 14 15 31 32 33 34 35 36];
i2=[4 5 6 7 8 9 10 11 12];
c2=[1 2 3 13 14 15 16 17 18 19 20 21];
%
% The following is the mapping matrix.
% The mapping matrix is not general and is case specific
%
M=[eye(12);
   0 0 0 0 0 0 -1 0 0 0 0 0;
   0 0 0 0 0 0 0 -1 0 0 0 0;
   0 0 0 0 0 0 0 0 -1 0 0 0;
   0 0 0 -1 0 0 0 0 0 0 0 0;
   0 0 0 0 -1 0 0 0 0 0 0 0;
   0 0 0 0 0 -1 0 0 0 0 0 0;
   0 0 0 0 0 0 0 0 0 -1 0 0;
   0 0 0 0 0 0 0 0 0 0 -1 0;
   0 0 0 0 0 0 0 0 0 0 0 -1;
   -1 0 0 0 0 0 0 0 0 0 0 0;
   0 -1 0 0 0 0 0 0 0 0 0 0;
   0 0 -1 0 0 0 0 0 0 0 0 0];
%
k= 25;      % spring constant
B=.02;      % viscous damping constant
```

APPENDIX F MATLAB CODE FOR EXAMPLE SIX

```
clear;
clg;
%                              finesprdamp7
%
% This program will calculate the eigenvalues, eigenvectors
% natural frequencies, and frequency response function matrix
% for a three degree of freedom at each node element.
% The system is modeled with beam elements that are
% aligned in the same plane but at any angle (2-D).
%          •--------•
%          |        |
%          |        |
%          •--------•
%          |        |
%          |        |
%          •--------•
% This program works for a beam element modeled with six general
% coordinates and thus six DOF.
%          (|-•--------•-|)
%
% the user must enter the following data to meet the beam configuration
% (E) youngs modulus psi
% (I) area moment of inertia in^4
% (WTD) weight density lbf/in^3
% (A) cross sectional area in^2
% conductivity [ the node connection mapping ]
% node coordinates [ cartesian coordinates for each node ]
% spring-damper conductivity
% (k) spring constant
% (q) viscous frequency dependent damping coefficient
%
% call the data file
minihullsprdam_d7
%
% calculate the number of beam elements
t0=clock;
flops(0);
a=size(con);
numel=a(1);
%
% calculate the number of nodes.
b=size(coord);
nodes=b(1);
%
```

```
% convert the coordinates in to the correct units  in.
coord=coord*12;
%
% calculate the beam element lengths and beam angles in radians
for i=1:numel
IC=con(i,1);
ID=con(i,2);
l(1,i)=sqrt((coord(ID,1)-coord(IC,1))^2+(coord(ID,2)-coord(IC,2))^2);
DX(i)=coord(ID,1)-coord(IC,1);
DY(i)=coord(ID,2)-coord(IC,2);
  if DX(i)>=0 & DY(i)>=0;
     t(1,i)=acos(DX(i)/l(1,i));
  elseif DX(i)<0 & DY(i)>=0;
     t(1,i)=acos(DY(i)/l(1,i))+pi/2;
  elseif DX(i)<0 & DY(i)<=0;
     t(1,i)=acos(abs(DX(i))/l(1,i))+pi;
  else
     t(1,i)=acos(abs(DY(i))/l(1,i))+(3*pi/2);
  end;
end;
%
% call trig function
[c,s]=ftrig(t,numel);
%
% calculate radius of gyration
for i=1:numel
r(1,i)=sqrt(I(i)/A(i));
end;
%
% create the global matrix which is all zeroes.
%
kg=[zeros(nodes*3,nodes*3)];
mg=[zeros(nodes*3,nodes*3)];
%
% assemble the elemental matricies to the global matrix.
%
for i=1:numel
[kel,mel]=felement6(l(i),WTD(i),I(i),E(i),A(i),r(i),c(i),s(i));
%
v=con(i,1);
w=con(i,2);
%
kg(3*v-2:3*v,3*v-2:3*v) = kg(3*v-2:3*v,3*v-2:3*v) + kel(1:3,1:3);
kg(3*v-2:3*v,3*w-2:3*w) = kg(3*v-2:3*v,3*w-2:3*w) + kel(1:3,4:6);
kg(3*w-2:3*w,3*v-2:3*v) = kg(3*w-2:3*w,3*v-2:3*v) + kel(4:6,1:3);
kg(3*w-2:3*w,3*w-2:3*w) = kg(3*w-2:3*w,3*w-2:3*w) + kel(4:6,4:6);
%
mg(3*v-2:3*v,3*v-2:3*v) = mg(3*v-2:3*v,3*v-2:3*v) + mel(1:3,1:3);
mg(3*v-2:3*v,3*w-2:3*w) = mg(3*v-2:3*v,3*w-2:3*w) + mel(1:3,4:6);
mg(3*w-2:3*w,3*v-2:3*v) = mg(3*w-2:3*w,3*v-2:3*v) + mel(4:6,1:3);
mg(3*w-2:3*w,3*w-2:3*w) = mg(3*w-2:3*w,3*w-2:3*w) + mel(4:6,4:6);
```

```
end
%
% This section will connect a spring-damper system to the global
% stiffness matrix. The spring-damper system is made up of a set of
% three springs and dampers that correspond to the degrees of freedom
% at node. It attaches to the global stiffness matrix based on the
% the spring-damper connectivity.
%
kgds=kg;
d = size(scon);
numspg = d(1);
count=0;
for Comega=.1:.1:25
count=count+1;
for j=1:numspg;
[kdsprg]=fsprngdampC(k(j),Comega,q(j));
%
x=scon(j,1);
y=scon(j,2);
%
kgds(3*x-2:3*x,3*x-2:3*x) = kg(3*x-2:3*x,3*x-2:3*x) + kdsprg(1:3,1:3);
kgds(3*x-2:3*x,3*y-2:3*y) = kg(3*x-2:3*x,3*y-2:3*y) + kdsprg(1:3,4:6);
kgds(3*y-2:3*y,3*x-2:3*x) = kg(3*y-2:3*y,3*x-2:3*x) + kdsprg(4:6,1:3);
kgds(3*y-2:3*y,3*y-2:3*y) = kg(3*y-2:3*y,3*y-2:3*y) + kdsprg(4:6,4:6);
end
%
% apply the boundary conditions
% the user must adjust the global matrix to meet the boundary conditions
%
% to delete rows
kgds([BC],:) = [];
mg([BC],:) = [];
% to delete columns
kgds(:,[BC]) = [];
mg(:,[BC]) = [];
%
% call the function and calculate eigenvectors and the eigenvalues
% which are the mode shapes and natural frequency in (rad/sec)^2
%
[lambda,phi]=fgmodes(kgds,mg);
% now convert the eigenvalues to nat frequency in (rad/sec) and
% hertz(1/sec)
%
omega = sqrt(lambda);
freq = omega/(2*pi);

Z=kgds-Comega^2*mg;
H=inv(Z);
%
HH(count)=H(8,8);
end;
```

```
etime(clock,t0),flops
Comega=.1:.1:25;
Freq=Comega/(2*pi);
plot(Freq,20*log(HH)),grid
xlabel('Frequency Hz')
ylabel('FRF at coordinate of interest dB')
%
% END


%              minihullsprdam_d7
%
%
% This is the data for the finite element program with three
% degrees of freedom at a node.
%
%       (|-•---------•-|)
%
% the data will be in the form of
% - (E) youngs modulus psi
% - (I) area moment of inertia in^4
% - (WTD) weight density lbf/in^3
% - (A) cross sectional area in^2
% - conductivity [ the node connection mapping ]
% - node coordinates [ cartesian coords for each node in ft. ]
%                      the main program will convert to in..
% - (q) frequency dependent viscous damping coefficient
% - (k) spring constant
E=[30 30 30 30 30 30 30 30 30 30 30 30 30 30 30 30 30 30]*1e6;
I=[.02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083
.02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083];%1/12bh^3 b=2,
h=.5
A=[1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1];
WTD=[.2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832
.2832 .2832 .2832 .2832 .2832 .2832 .2832];
q=[.1 .1 .1 .1];                    % damping coefficient
k=[25 25 25 25];                    % lbs/in
%
con=[1,2;
     2,3;
     3,4;
     4,5;
     5,6;
     6,7;
     7,8;
     8,9;
     9,10;
     10,11;
     11,12;
     12,1;
     13,14;
```

```
      14,15;
      15,16;
      16,17;
      14,18;
      16,19];
%
% spring connectivity
scon=[11,13;
      17,5;
      12,18;
      19,4];
%
coord=[4,0;
       8,0;
       12,0;
       16,4;
       16,8;
       16,12;
       12,16;
       8,16;
       4,16;
       0,12;
       0,8;
       0,4;
       0,8;
       4,8;
       8,8;
       12,8;
       16,8;
       0,4;
       16,4];
%
BC=[];


%              out_struc7
%
%
% This is the data for the finite element program with three
% degrees of freedom at a node.
%
%       (|-•---------•-|)
%
% the data will be in the form of
% - (E) youngs modulus psi
% - (I) area moment of inertia in^4
% - (WTD) weight density lbf/in^3
% - (A) cross sectional area in^2
% - conductivity [ the node connection mapping ]
% - node coordinates [ cartesian coords for each node in ft. ]
%                      the main program will convert to in..
```

```
%
E=[30 30 30 30 30 30 30 30 30 30 30 30]*1e6;
I=[ .02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083 .02083
.02083 .02083 .02083];%1/12bh^3 b=2, h=.5
A=[ 1 1 1 1 1 1 1 1 1 1 1 1];
WTD=[.2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832 .2832
.2832];
%
con=[1,2;
     2,3;
     3,4;
     4,5;
     5,6;
     6,7;
     7,8;
     8,9;
     9,10;
     10,11;
     11,12;
     12,1];
%
coord=[4,0;
       8,0;
       12,0;
       16,4;
       16,8;
       16,12;
       12,16;
       8,16;
       4,16;
       0,12;
       0,8;
       0,4];
%
BC=[];


%              inner_struc7
%
%
% This is the data for the finite element program with three
% degrees of freedom at a node.
%
%       (|-•---------•-|)
%
% the data will be in the form of
% - (E) youngs modulus psi
% - (I) area moment of inertia in^4
% - (WTD) weight density lbf/in^3
% - (A) cross sectional area in^2
% - conductivity [ the node connection mapping ]
```

```
% - node coordinates ( cartesian coords for each node in ft.
%                      the main program will convert to in..
%
E=[30 30 30 30 30 30]*1e6;
I=[ .02083 .02083 .02083 .02083 .02083 .02083];%1/12bh^3 b=2, h=.5
A=[ 1 1 1 1 1 1];
WTD=[ .2832 .2832 .2832 .2832 .2832 .2832];

con=[1,2;
     2,3;
     3,4;
     4,5;
     2,6;
     4,7];
%
coord=[0,4;
       4,4;
       8,4;
       12,4;
       16,4;
       0,0;
       16,0];
%
BC=[];


clear;
clg;
%
%                  FRF_INDSPRDAM_7
%
% Load data from running unifineel program for each substructure.
% The K and M matrix for each substructure is saved.
%
load ex7a.mat    % k1 m1 are stored here
load ex7b.mat    % k2 m2 are stored here
%
%  We need to create a single FRF matrix representing
%  both substructures in the form:
%
%      [hee] =  [ h(i,i)   h(i,c) ]
%               [ h(c,i)   h(c,c) ]
%
%
%  So we create arrays containing the DOF numbers of our original
%  models which correspond the the "c" and "i" coordinates for
%  each substructure.
%
% call the synthesis data file in now which contains the
% internal coordinates and connection coordinates for each sub
% structure.
```

```
%
% i1= internal coords of sub structure 1
% i2= internal coords of sub structure 2
% c1= connection coords of sub structure 1
% c2= connection coords of sub structure 2
%
FRF_Indsprdam_data7
%
t0 = clock;
flops(0);
count=0;
for Comega=.1:.1:50
count=count+1;
%
%           Form Frequency Response Models for Each Substructure
%           ~~~~ ~~~~~~~~~ ~~~~~~~~ ~~~~~~ ~~~ ~~~~ ~~~~~~~~~~~~
%
z1=k1-Comega^2*m1;
z2=k2-Comega^2*m2;
h1=inv(z1);
h2=inv(z2);
%
a=size(i1);
b=size(c1);
c=size(i2);
d=size(c2);
%
aa=a(2);
bb=b(2);
cc=c(2);
dd=d(2);
%
%   Remember, we are trying to calculate the following:
%
%   hee* = hee - hec * M * inv(zr + hccr ) * M' * hce
%   hccr = M' * hcc * M
%   zr = pinv(M) * z * pinv(M') which is just the identity matrix size
%                          3 times the number of spring-damp systems
%
%  So we need to assemble [hee], [hec], [hce] and [hcc] using the
%  the coordinate sets we just defined.
%  These matrices contain the FRF data for both substructures
%  prior to coupling, i.e the pre-synthesis FRF data.
%
%                    Coordinate Partitioning
%                    ~~~~~~~~~~ ~~~~~~~~~~~~
%
%  Build up uncoupled FRF matrix and sub-partitions:
%
hee = [h1(i1,i1)  zeros(aa,cc)   h1(i1,c1)  zeros(aa,dd);
      zeros(cc,aa)   h2(i2,i2)  zeros(cc,bb)  h2(i2,c2);
```

```
       h1(c1,i1)    zeros(bb,cc)    h1(c1,c1)   zeros(bb,dd);
       zeros(dd,aa)  h2(c2,i2)  zeros(dd,bb)  h2(c2,c2)];
%
hec = [h1(i1,c1)  zeros(aa,dd);
       zeros(cc,bb)  h2(i2,c2);
       h1(c1,c1)    zeros(bb,dd);
       zeros(dd,bb)  h2(c2,c2)];
%
hcc =  [h1(c1,c1) zeros(bb,dd);
        zeros(dd,bb)  h2(c2,c2)];
%
hce=   [h1(c1,i1)   zeros(bb,cc)   h1(c1,c1)  zeros(bb,dd);
       zeros(dd,aa)  h2(c2,i2)  zeros(dd,bb)  h2(c2,c2)];
%
%  We can now perform the synthesis:
zr = (k + j*Comega*k*exp(-q*Comega)) * eye(12);
hccr = M' * hcc * M;
heestar = hee - hec * M * inv(inv(zr) + hccr ) * M' * hce;
HH(count)=heestar(8,8);
end;
etime(clock,t0),flops
%
Comega=.1:.1:50;
Freq=Comega/(2*pi);
plot(Freq,20*log(HH)),grid
xlabel(' Frequency Hz ')
ylabel('FRF at coordinate of interest dB ')


%                    FRF_Indsprdam_data7
%
% This is the data file for the synthesis program.
% The following data will be provided by this file.
%
% i1- internal coords sub structure 1
% i2- internal coords sub structure 2
% c1- connection coords sub structure 1
% c2- connection coords sub structure 2
%
i1=[1 2 3 4 5 6 7 8 9 16 17 18 19 20 21 22 23 24 25 26 27 28 29 30];
c1=[10 11 12 13 14 15 31 32 33 34 35 36];
i2=[4 5 6 7 8 9 10 11 12];
c2=[1 2 3 13 14 15 16 17 18 19 20 21];
%
% the following is the mapping matrix
% the mapping matrix is not general and is
% case specific
%
M=[eye(12);
   0 0 0 0 0 0 -1 0 0 0 0 0;
   0 0 0 0 0 0 0 -1 0 0 0 0;
```

```
   0 0 0 0 0 0 0 0 -1 0 0 0;
   0 0 0 -1 0 0 0 0 0 0 0 0;
   0 0 0 0 -1 0 0 0 0 0 0 0;
   0 0 0 0 0 -1 0 0 0 0 0 0;
   0 0 0 0 0 0 0 0 0 -1 0 0;
   0 0 0 0 0 0 0 0 0 0 -1 0;
   0 0 0 0 0 0 0 0 0 0 0 -1;
   -1 0 0 0 0 0 0 0 0 0 0 0;
   0 -1 0 0 0 0 0 0 0 0 0 0;
   0 0 -1 0 0 0 0 0 0 0 0 0];
%
% spring constant and damping coefficient
k= 25;
q=.1;
```

APPENDIX G MATLAB CODE FOR EXAMPLE SEVEN

```
clear

%                          unifineelstress
%
% This program will calculate the eigenvalues, eigenvectors
% natural frequencies, and stress frequency response function matrix
% for a three degree of freedom at each node element.
% The system is modeled with beam elements that are
% aligned in the same plane but at any angle (2-D).
%          •--------•
%          |        |
%          |        |
%          •--------•
%          |        |
%          |        |
%          •--------•
% This program works for a beam element modeled with six general
% coordinates and thus six DOF.
%          (|-•-------•-|)
%
% the user must enter the following data to meet the beam configuration
% (E) youngs modulus psi
% (I) area moment of inertia in^4
% (WTD) weight density lbf/in^3
% (A) cross sectional area in^2
% conductivity [ the node connection mapping ]
% node coordinates [ cartesian coordinates for each node ]
% (beam)the beam element of interest
% (cc) structure connection coordinates
% (bcoord) beam element coordinates
% (cd) distance from beam center to outer most fiber
clear;
% call the data file
hullstressdata
%
% calculate the number of beam elements
a=size(con);
numel=a(1);
%
% calculate the number of nodes.
b=size(coord);
nodes=b(1);
%
% convert the coordinates in to the correct units (in.)
```

```
%coord=coord*12;
%
% calculate the beam element lengths and beam angles in radians
for i=1:numel
IC=con(i,1);
ID=con(i,2);
l(1,i)=sqrt((coord(ID,1)-coord(IC,1))^2+(coord(ID,2)-coord(IC,2))^2);
DX(i)=coord(ID,1)-coord(IC,1);
DY(i)=coord(ID,2)-coord(IC,2);
  if DX(i)>=0 & DY(i)>=0;
     t(1,i)=acos(DX(i)/l(1,i));
  elseif DX(i)<0 & DY(i)>=0;
     t(1,i)=acos(DY(i)/l(1,i))+pi/2;
  elseif DX(i)<0 & DY(i)<=0;
     t(1,i)=acos(abs(DX(i))/l(1,i))+pi;
  else
     t(1,i)=acos(abs(DY(i))/l(1,i))+(3*pi/2);
  end;
end;
%
% call trig function
[c,s]=ftrig(t,numel);
%
% calculate radius of gyration
for i=1:numel
r(1,i)=sqrt(I(i)/A(i));
end;
%
% create the global matrix which is all zeroes.
%
kg=[zeros(nodes*3,nodes*3)];
mg=[zeros(nodes*3,nodes*3)];
%
% assemble the elemental matricies to the global matrix.
%
for i=1:numel
[kel,mel]=felement6(l(i),WTD(i),I(i),E(i),A(i),r(i),c(i),s(i));
%
v=con(i,1);
w=con(i,2);
%
kg(3*v-2:3*v,3*v-2:3*v) = kg(3*v-2:3*v,3*v-2:3*v) + kel(1:3,1:3);
kg(3*v-2:3*v,3*w-2:3*w) = kg(3*v-2:3*v,3*w-2:3*w) + kel(1:3,4:6);
kg(3*w-2:3*w,3*v-2:3*v) = kg(3*w-2:3*w,3*v-2:3*v) + kel(4:6,1:3);
kg(3*w-2:3*w,3*w-2:3*w) = kg(3*w-2:3*w,3*w-2:3*w) + kel(4:6,4:6);
%
mg(3*v-2:3*v,3*v-2:3*v) = mg(3*v-2:3*v,3*v-2:3*v) + mel(1:3,1:3);
mg(3*v-2:3*v,3*w-2:3*w) = mg(3*v-2:3*v,3*w-2:3*w) + mel(1:3,4:6);
mg(3*w-2:3*w,3*v-2:3*v) = mg(3*w-2:3*w,3*v-2:3*v) + mel(4:6,1:3);
mg(3*w-2:3*w,3*w-2:3*w) = mg(3*w-2:3*w,3*w-2:3*w) + mel(4:6,4:6);
end
```

```
%
% apply the boundary conditions
% the user must adjust the global matrix to meet the boundary conditions
%
% to delete rows
kg([BC],:) = [];
mg([BC],:) = [];
% to delete columns
kg(:,[BC]) = [];
mg(:,[BC]) = [];
%
% call the function and calculate eigenvectors and the eigenvalues
% which are the mode shapes and natural frequency in (rad/sec)^2
%
[lambda,phi]=fgmodes(kg,mg);
%
% this converts the eigenvalues to nat frequency in (rad/sec) and
% and hertz(1/sec)
%
omega = sqrt(lambda);
freq = omega/(2*pi);
%
% calculate the FRF over the freq. of interest
%
count=0;
for Comega=2:1:1600
count=count+1;
Z=kg-Comega^2*mg;
H=inv(Z);
% save the coordinate of interest to plot
HH(count)=H(1,1);
%
% this portion calculates the stress FRF in a given element
%
HCOL=size(cc);
NHCOL=HCOL(2);
MEQ=[0 l(beam)/2 1 0 0 0];
[Trmatrix]=ftrans(c,s,beam);
[kel]=fkelement6(l(beam),WTD(beam),I(beam),E(beam),A(beam),r(beam));
for i=1:NHCOL
HEL=H(:,cc(i));
HELR=HEL(bcoord,:);
HLOCAL=Trmatrix*HELR;
NODEF=kel*HLOCAL;
stress(1,i)=cd/I(beam)*MEQ*NODEF;
end;
Stress(count)=stress(1,3);
% this is the end of the stress FRF calculation
end;
Comega=2:1:1600;
Freq=Comega/(2*pi);
```

```
plot(Freq,20*log(HH)),grid,pause
plot(Freq,20*log(Stress)),grid
%
% end


%              hullstressdata
%
%
% This is the data for the finite element program.
%
% the data will be in the form of
% - (E) youngs modulus psi
% - (I) area moment of inertia in^4
% - (WTD) weight density lbf/in^3
% - (A) cross sectional area in^2
% - conductivity [ the node connection mapping ]
% - node coordinates [ cartesian coords for each node in ft. ]
%                      the main program will convert to in..
% - (beam) the beam element of interest
% - (bcoord) the beam element coordinates
% - (cc) structure connection coordinates
% - (cd) distance from beam center to outer most fiber in
%
E=[30 30 30 30 30 ]*1e6;
I=[.1666 .1666 .1666 .1666 .1666]*1e-3;%1/12bh^3 b=2, h=0.1
A=[.2 .2 .2 .2 .2];
WTD=[.2832 .2832 .2832 .2832 .2832];
beam=4;
cc=[4 5 6 10 11 12];
bcoord=[1 2 3 10 11 12];
cd=.05;
%
con=[1,2;
     2,3;
     3,4;
     4,1;
     4,2];
%
coord=[10,0;
       20,10;
       10,20;
       0,10];  % this is in (in.)
%
BC=[];


%              hullstressdata1
%
%
% This is the data for the finite element program.
```

```
% This is the outer structure.
%
% the data will be in the form of
% - (E) youngs modulus psi
% - (I) area moment of inertia in^4
% - (WTD) weight density lbf/in^3
% - (A) cross sectional area in^2
% - conductivity [ the node connection mapping ]
% - node coordinates [ cartesian coords for each node in ft. ]
%                      the main program will convert to in..
%
E=[30 30 30 30]*1e6;
I=[.1666 .1666 .1666 .1666]*1e-3;%1/12bh^3 b=2, h=0.1
A=[.2 .2 .2 .2];
WTD=[.2832 .2832 .2832 .2832];
%
con=[1,2;
     2,3;
     3,4;
     4,1];
%
coord=[10,0;
       20,10;
       10,20;
       0,10];  % this is in (in.)
%
BC=[];


%              hullstressdata2
%
%
% This is the data for the finite element program.
% This is the inner structure.
%
% the data will be in the form of
% - (E) youngs modulus psi
% - (I) area moment of inertia in^4
% - (WTD) weight density lbf/in^3
% - (A) cross sectional area in^2
% - conductivity [ the node connection mapping ]
% - node coordinates [ cartesian coords for each node in ft. ]
%                      the main program will convert to in..
%
E=[30]*1e6;
I=[.1666 ]*1e-3;%1/12bh^3 b=2, h=0.1
A=[.2];
WTD=[.2832];
%
con=[1,2];
%
```

```
coord=[20,0;
       0,0];   % this is in in.
%
BC=[];


%                    program FRF_stress8
%
% this program will calculate stress FRF by the indirect
% coupling method
%
clear;
clg;
%load ex8a.mat    % k1, m1 are stored here
%load ex8b.mat    % k2, m2 are stored here
%
stressdata8
%
% We need to partion the H matrix of the structure to
% be modified in the following way.
%
%     [ ii | ic ]
%     -----------
%     [ ci | cc ]
%
%     [hee] = [ ii ic
%               ci cc ]
%
%     [hec] = [ ic
%               cc ]
%
%     [hcc] = [ cc ]
%
% define the variables
% internal moment equation in vector form
MEQ=[0 11/2 1 0 0 0];
% number of columns in the frf matrix
nhcol=nodes*3;
% calling functions
[kel]=frfkelement6(ll,WTD,I,E,A,rr);
[Trmatrix]=frftrans(ccc,ss)
% set the clock and flops
t0=clock;
flops(0);
count=0;
for Comega=2:1:1600
count=count+1;
%
%           Form Frequency Response Models for Each Substructure
%           ---- --------- -------- ------ --- ---- -----------
```

```
z1=k1-Comega^2*m1;
z=k2-Comega^2*m2;
h=inv(z1);
%
%                    Coordinate Partitioning
%                    ---------- -------------
%
hcc = [h(cc,cc)];
%
hce = [h(cc,ic) h(cc,cc)];
%
% calculate the stress of desired beam
for i=1:nhcol
hel=h(:,i);
helr=hel(bcoord,:);
hlocal=Trmatrix*helr;
nodalF=kel*hlocal;
S(1,i)=cd/I*MEQ*nodalF;
end;
hse=[S(1,ic) S(1,cc)];
hsc=[S(1,cc)];
% now we will synthesize the stresses
hsestar=hse - hsc * inv(inv(z) + hcc) * hce;
HS(count) = hsestar(1,9);     % this is for coord 6
end
etime(clock,t0),flops
Comega=2:1:1600;
Freq=Comega/(2*pi);
plot(Freq,20*log(HS)),grid
xlabel(' Frequency Hz ')
ylabel('Stress FRF at coordinate  of interest dB ')


%                          stressdata8
%
% This is the data file for the stress synthesis program.
% The following data will be provided by this file.
%
% ic- internal coords of synthesized structure
% cc- connection coords of synthesized structure
%
ic=[1 2 3 7 8 9];
cc=[4 5 6 10 11 12];
bcoord=[1 2 3 10 11 12];
%
E=[30]*1e6;
I=[.1666 ]*1e-3;%1/12bh^3 b=2, h=0.1
A=[.2];
WTD=[.2832];
cd=.05;
nodes=4;
```

APPENDIX H MATLAB CODE FOR EXAMPLE EIGHT

```
clear;
clg;
%
%                          unifineel1
%
% This program will calculate the eigenvalues, eigenvectors
% natural frequencies, and frequency response function matrix
% for a three degree of freedom at each node element.
% The system is modeled with beam elements that are
% aligned in the same plane but at any angle (2-D).
%          •--------•
%          |        |
%          |        |
%          •--------•
%          |        |
%          |        |
%          •--------•
% This program works for a beam element modeled with six general
% coordinates and thus six DOF.
%          (|-•-------•-|)
%
% the user must enter the following data to meet the beam configuration
% (E) youngs modulus psi
% (I) area moment of inertia in^4
% (WTD) weight density lbf/in^3
% (A) cross sectional area in^2
% conductivity [ the node connection mapping ]
% node coordinates [ cartesian coordinates for each node ]
% (bb) structural proportional damping constant
%
% call the data file
fxbeam_data9
%
% start the program clock and flops to determine program running
% time and floating point calculations
t0=clock;
flops(0);
%
% calculate the number of beam elements
a=size(con);
numel=a(1);
%
% calculate the number of beam elements proportionally damped
aa=size(dcon);
```

```
numel_damp=aa(1);
%
% calculate the number of nodes.
b=size(coord);
nodes=b(1);
%
% convert the coordinates in to the correct units (in.)
coord=coord*12;
%
% calculate the beam element lengths and beam angles in radians
for i=1:numel
IC=con(i,1);
ID=con(i,2);
l(1,i)=sqrt((coord(ID,1)-coord(IC,1))^2+(coord(ID,2)-coord(IC,2))^2);
DX(i)=coord(ID,1)-coord(IC,1);
DY(i)=coord(ID,2)-coord(IC,2);
  if DX(i)>=0 & DY(i)>=0;
     t(1,i)=acos(DX(i)/l(1,i));
  elseif DX(i)<0 & DY(i)>=0;
     t(1,i)=acos(DY(i)/l(1,i))+pi/2;
  elseif DX(i)<0 & DY(i)<=0;
     t(1,i)=acos(abs(DX(i))/l(1,i))+pi;
  else
     t(1,i)=acos(abs(DY(i))/l(1,i))+(3*pi/2);
  end;
end;
%
% call trig function
[c,s]=ftrig(t,numel);
%
% calculate radius of gyration
for i=1:numel
r(1,i)=sqrt(I(i)/A(i));
end;
%
% create the global matrix which is all zeroes.
%
kg=[zeros(nodes*3,nodes*3)];
mg=[zeros(nodes*3,nodes*3)];
%
% assemble the elemental matricies to the global matrix.
%
for i=1:numel
[kel,mel]=felement6(l(i),WTD(i),I(i),E(i),A(i),r(i),c(i),s(i));
%
v=con(i,1);
w=con(i,2);
%
kg(3*v-2:3*v,3*v-2:3*v) = kg(3*v-2:3*v,3*v-2:3*v) + kel(1:3,1:3);
kg(3*v-2:3*v,3*w-2:3*w) = kg(3*v-2:3*v,3*w-2:3*w) + kel(1:3,4:6);
kg(3*w-2:3*w,3*v-2:3*v) = kg(3*w-2:3*w,3*v-2:3*v) + kel(4:6,1:3);
```

```
kg(3*w-2:3*w,3*w-2:3*w  = kg(3*w-2:3*w,3*w-2:3*w  + kel(4:6,4:6 ;
%
mg(3*v-2:3*v,3*v-2:3*v) = mg(3*v-2:3*v,3*v-2:3*v) + mel(1:3,1:3);
mg(3*v-2:3*v,3*w-2:3*w) = mg(3*v-2:3*v,3*w-2:3*w) + mel(1:3,4:6);
mg(3*w-2:3*w,3*v-2:3*v) = mg(3*w-2:3*w,3*v-2:3*v) + mel(4:6,1:3);
mg(3*w-2:3*w,3*w-2:3*w) = mg(3*w-2:3*w,3*w-2:3*w) + mel(4:6,4:6);
end
%
% apply structural prop. damping to the k matrix and set global
% k matrix to equal damped matrix
%
% calculate the beam element lengths and beam angles in radians
% for the damped beams
for i=1:numel_damp
IC=dcon(i,1);
ID=dcon(i,2);
l(1,i)=sqrt((coord(ID,1)-coord(IC,i))^2+(coord(ID,2)-coord(IC,2))^2);
DX(i)=coord(ID,1)-coord(IC,1);
DY(i)=coord(ID,2)-coord(IC,2);
  if DX(i)>=0 & DY(i)>=0;
     t(1,i)=acos(DX(i)/l(1,i));
  elseif DX(i)<0 & DY(i)>=0;
     t(1,i)=acos(DY(i)/l(1,i))+pi/2;
  elseif DX(i)<0 & DY(i)<=0;
     t(1,i)=acos(abs(DX(i))/l(1,i))+pi;
  else
     t(1,i)=acos(abs(DY(i))/l(1,i))+(3*pi/2);
  end;
end;
%
% call trig function
[c,s]=ftrig(t,numel_damp);
%
% calculate radius of gyration
for i=1:numel_damp
r(1,i)=sqrt(I(i)/A(i));
end;
%
kgd=kg;
for u=1:numel_damp
[kel]=felement6(l(u),WTD(u),I(u),E(u),A(u),r(u),c(u),s(u));
%
v=dcon(u,1);
w=dcon(u,2);
%
kgd(3*v-2:3*v,3*v-2:3*v) = kgd(3*v-2:3*v,3*v-2:3*v) + j*bb*kel(1:3,1:3);
kgd(3*v-2:3*v,3*w-2:3*w) = kgd(3*v-2:3*v,3*w-2:3*w) + j*bb*kel(1:3,4:6);
kgd(3*w-2:3*w,3*v-2:3*v) = kgd(3*w-2:3*w,3*v-2:3*v) + j*bb*kel(4:6,1:3);
kgd(3*w-2:3*w,3*w-2:3*w) = kgd(3*w-2:3*w,3*w-2:3*w) + j*bb*kel(4:6,4:6);
%
end
```

```
%
% now apply the boundary conditions
% the user must adjust the global matrix to meet the boundary conditions
%
% to delete rows
kg([BC],:) = [];
kgd([BC],:) = [];
mg([BC],:) = [];
% to delete columns
kg(:,[BC]) = [];
kgd(:,[BC]) = [];
mg(:,[BC]) = [];
%
% call the function and calculate eigenvectors and the eigenvalues
% which are the mode shapes and undamped natural frequency in
(rad/sec)^2
%
[lambda,phi]=fgmodes(kg,mg);
% this now converts the eigenvalues to nat frequency in (rad/sec) and
% hertz(1/sec)
%
omega = sqrt(lambda);
freq = omega/(2*pi);
%
% constuct the frequency response plot over the frequencies
% of interest
count=0;
for Comega=.1:.5:500
count=count+1;
Z=kgd-Comega^2*mg;
H=inv(Z);
%
% this determines the coordinate of intrest to plot
HH(count)=H(2,6);
end;
% end the program clock and flops
etime(clock,t0),flops
Comega=.1:.5:500;
Freq=Comega/(2*pi);
plot(Freq,20*log(HH)),grid
xlabel(' Frequency Hz ')
ylabel('FRF at coordinate of interest dB ')
% end


clear;
clg;

%                          unifineellmode
%
% This program will calculate the eigenvalues, eigenvectors
```

```
% natural frequencies, and frequency response function matrix
% for a three degree of freedom at each node element using modal
% representation. The system is modeled with beam elements that are
% aligned in the same plane but at any angle (2-D).
%         •--------•
%         |        |
%         |        |
%         •--------•
%         |        |
%         |        |
%         •--------•
% This program works for a beam element modeled with six general
% coordinates and thus six DOF.
%         (|-•--------•-|)
%
% the user must enter the following data to meet the beam configuration
% (E) youngs modulus psi
% (I) area moment of inertia in^4
% (WTD) weight density lbf/in^3
% (A) cross sectional area in^2
% conductivity [ the node connection mapping ]
% node coordinates [ cartesian coordinates for each node ]
% (bb) structural proportional damping constant
%
% call the data file
fxbeam_data9
%
% start the program clock and flops to determine program running
% time and floating point calculations
t0=clock;
flops(0);
%
% calculate the number of beam elements
a=size(con);
numel=a(1);
%
% calculate the number of beam elements proportionally damped
aa=size(dcon);
numel_damp=aa(1);
%
% calculate the number of nodes.
b=size(coord);
nodes=b(1);
%
% convert the coordinates in to the correct units (in.)
coord=coord*12;
%
% calculate the beam element lengths and beam angles in radians
for i=1:numel
IC=con(i,1);
ID=con(i,2);
```

```
l(1,i)=sqrt((coord(ID,1)-coord(IC,1))^2+(coord(ID,2)-coord(IC,2))^2);
DX(i)=coord(ID,1)-coord(IC,1);
DY(i)=coord(ID,2)-coord(IC,2);
  if DX(i)>=0 & DY(i)>=0;
     t(1,i)=acos(DX(i)/l(1,i));
  elseif DX(i)<0 & DY(i)>=0;
     t(1,i)=acos(DY(i)/l(1,i))+pi/2;
  elseif DX(i)<0 & DY(i)<=0;
     t(1,i)=acos(abs(DX(i))/l(1,i))+pi;
  else
     t(1,i)=acos(abs(DY(i))/l(1,i))+(3*pi/2);
  end;
end;
%
% call trig function
[c,s]=ftrig(t,numel);
%
% calculate radius of gyration
for i=1:numel
r(1,i)=sqrt(I(i)/A(i));
end;
%
% create the global matrix which is all zeroes.
%
kg=[zeros(nodes*3,nodes*3)];
mg=[zeros(nodes*3,nodes*3)];
%
% assemble the elemental matricies to the global matrix.
%
for i=1:numel
[kel,mel]=felement6(l(i),WTD(i),I(i),E(i),A(i),r(i),c(i),s(i));
%
v=con(i,1);
w=con(i,2);
%
kg(3*v-2:3*v,3*v-2:3*v) = kg(3*v-2:3*v,3*v-2:3*v) + kel(1:3,1:3);
kg(3*v-2:3*v,3*w-2:3*w) = kg(3*v-2:3*v,3*w-2:3*w) + kel(1:3,4:6);
kg(3*w-2:3*w,3*v-2:3*v) = kg(3*w-2:3*w,3*v-2:3*v) + kel(4:6,1:3);
kg(3*w-2:3*w,3*w-2:3*w) = kg(3*w-2:3*w,3*w-2:3*w) + kel(4:6,4:6);
%
mg(3*v-2:3*v,3*v-2:3*v) = mg(3*v-2:3*v,3*v-2:3*v) + mel(1:3,1:3);
mg(3*v-2:3*v,3*w-2:3*w) = mg(3*v-2:3*v,3*w-2:3*w) + mel(1:3,4:6);
mg(3*w-2:3*w,3*v-2:3*v) = mg(3*w-2:3*w,3*v-2:3*v) + mel(4:6,1:3);
mg(3*w-2:3*w,3*w-2:3*w) = mg(3*w-2:3*w,3*w-2:3*w) + mel(4:6,4:6);
end
%
%
% apply structural prop. damping to the k matrix and set global
% k matrix to equal damped matrix
%
%
```

```
% calculate the beam element lengths and beam angles in radians
% for the damped beams
for i=1:numel_damp
IC=dcon(i,1);
ID=dcon(i,2);
l(1,i)=sqrt((coord(ID,1)-coord(IC,1))^2+(coord(ID,2)-coord(IC,2))^2);
DX(i)=coord(ID,1)-coord(IC,1);
DY(i)=coord(ID,2)-coord(IC,2);
  if DX(i)>=0 & DY(i)>=0;
     t(1,i)=acos(DX(i)/l(1,i));
  elseif DX(i)<0 & DY(i)>=0;
     t(1,i)=acos(DY(i)/l(1,i))+pi/2;
  elseif DX(i)<0 & DY(i)<=0;
     t(1,i)=acos(abs(DX(i))/l(1,i))+pi;
  else
     t(1,i)=acos(abs(DY(i))/l(1,i))+(3*pi/2);
  end;
end;
%
% call trig function
[c,s]=ftrig(t,numel_damp);
%
% calculate radius of gyration
for i=1:numel_damp
r(1,i)=sqrt(I(i)/A(i));
end;
%
kgd=kg;
for u=1:numel_damp
[kel]=felement6(l(u),WTD(u),I(u),E(u),A(u),r(u),c(u),s(u));
%
v=dcon(u,1);
w=dcon(u,2);
%
kgd(3*v-2:3*v,3*v-2:3*v) = kgd(3*v-2:3*v,3*v-2:3*v) + j*bb*kel(1:3,1:3);
kgd(3*v-2:3*v,3*w-2:3*w) = kgd(3*v-2:3*v,3*w-2:3*w) + j*bb*kel(1:3,4:6);
kgd(3*w-2:3*w,3*v-2:3*v) = kgd(3*w-2:3*w,3*v-2:3*v) + j*bb*kel(4:6,1:3);
kgd(3*w-2:3*w,3*w-2:3*w) = kgd(3*w-2:3*w,3*w-2:3*w) + j*bb*kel(4:6,4:6);
%
end
%
% apply the boundary conditions
% the user must adjust the global matrix to meet the boundary conditions
%
% to delete rows
kg([BC],:) = [];
kgd([BC],:) = [];
mg([BC],:) = [];
% to delete columns
kg(:,[BC]) = [];
kgd(:,[BC]) = [];
```

```
mg(:,[BC]) = [];
%
% call the function and calculate eigenvectors and the eigenvalues
% which are the mode shapes and undamped natural frequency in
(rad/sec)^2
%
[lambda,phi]=fgmodes(kg,mg);
% this now converts the eigenvalues to nat frequency in (rad/sec) and
% hertz(1/sec)
%
omega = sqrt(lambda);
freq = omega/(2*pi);
%
% constuct the frequency response plot over the frequencies
% of interest
count=0;
fs=size(freq);
nm=fs(1);
% extract the mode shapes for the coordinates of interest
phired=phi(ci,:);
for Comega=.1:.5:500
count=count+1;
% generate the diagonal frequency matrix
for i=1:nm
nomega(i)=1/(omega(i)^2-Comega^2);
end;
% generate the FRF matrix
H=phired*diag(nomega)*phired';
%
% this determines the coordinate of intrest to plot
HH(count)=H(1,3);     % this refers to (2,6)
end;
% end the program clock and flops
etime(clock,t0),flops
Comega=.1:.5:500;
Freq=Comega/(2*pi);
plot(Freq,20*log(HH)),grid
xlabel(' Frequency Hz ')
ylabel('FRF at coordinate of interest dB ')
% end


%              fxbeam_data9
%
%
% This is the data for the finite element program and the finite
% element program using modal representation.
%
% the data will be in the form of
% - (E) youngs modulus psi
% - (I) area moment of inertia in^4
```

```
% - (WTD) weight density lbf/in^3
% - (A) cross sectional area in^2
% - conductivity [ the node connection mapping ]
% - node coordinates [ cartesian coords for each node in ft. ]
%                      the main program will convert to in..
% - (bb) structural proportional damping constant
% - (ci) coordinates of interest
E=[30 30 30 30 ]*1e6;
I=[.02083 .02083 .02083 .02083];%1/12bh^3 b=2, h=.5
A=[1 1 1 1];
WTD=[.2832 .2832 .2832 .2832];
bb=0;
% coordinates of interest (any internal and all connection)
ci=[2 4 5 6];
con=[1,2;
     2,3;
     3,4;
     4,5];
%
dcon=[1,2;
     2,3;
     3,4;
     4,5];
%
coord=[0,0;
       4,0;
       8,0;
       12,0;
       16,0];
%
BC=[1 2 3 13 14 15];


%              right_struc9
%
%
% This is the data for the finite element program.
%
% the data will be in the form of
% - (E) youngs modulus psi
% - (I) area moment of inertia in^4
% - (WTD) weight density lbf/in^3
% - (A) cross sectional area in^2
% - conductivity [ the node connection mapping ]
% - node coordinates [ cartesian coords for each node in ft. ]
%                      the main program will convert to in..
% - (bb) structural proportional damping constant
%
E=[30 30]*1e6;
I=[ .02083 .02083];%1/12bh^3 b=2, h=.5
A=[ 1 1];
```

```
WTD=[.2832 .2832];
bb=0;
%
con=[1,2;
     2,3];
%
dcon=[1,2;
     2,3];
%
coord=[0,0;
       4,0;
       8,0];
%
BC=[7 8 9];


%              left_struc9
%
%
% This is the data for the finite element program.
%
% the data will be in the form of
% - (E) youngs modulus psi
% - (I) area moment of inertia in^4
% - (WTD) weight density lbf/in^3
% - (A) cross sectional area in^2
% - conductivity [ the node connection mapping ]
% - node coordinates [ cartesian coords for each node in ft. ]
%                      the main program will convert to in..
% - (bb) structural proportional damping constant
E=[30 30]*1e6;
I=[ .02083 .02083];%1/12bh^3 b=2, h=.5
A=[1 1];
WTD=[.2832 .2832];
bb=0;
con=[1,2;
     2,3];
%
dcon=[1,2;
     2,3];
%
coord=[0,0;
       4,0;
       8,0];
%
BC=[1 2 3];


clear
clg
%                    FRF_Synth9
```

```
%
% This program synthesizes by dynamic direct coupling using the boolean
% mapping matrix to synthesize two structures together.
%
% Load data from running unifineel program for each substructure.
% The K and M matrix for each structure is saved
load ex9a.mat     % k1 m1 is stored here
load ex9b.mat     % k2 m2 is stored here
% calculate the eigenvectors and eigenvalues for each structure
[lambda,phi]=fgmodes(k1,m1);
lambda1=lambda;
% calculate nat freq in rad/sec
omega1=sqrt(lambda1);
phi1=phi;
[lambda,phi]=fgmodes(k2,m2);
lambda2=lambda;
% calculate nat freq in rad/sec
omega2=sqrt(lambda2);
phi2=phi;
%
%  We need to create a single FRF matrix representing
%  both substructures in the form:

%      [hee] =  [ h(i,i)   h(i,c) ]
%               [ h(c,i)   h(c,c) ]
%
%  So we create arrays containing the DOF numbers of our original
%  models which correspond the the "c" and "i" coordinates for
%  each substructure.
%
% call the synthesis data file in now which contains the
% internal coordinates and connection coordinates for each sub
% structure.
% i1= internal coords of sub structure 1
% i2= internal coords of sub structure 2
% c1= connection coords of sub structure 1
% c2= connection coords of sub structure 2
% ci1= coordinates of interest of sub structure 1
% ci2= coordinates of interest of sub structure 2
% ni1= redefined internal coords of sub structure 1
% nc1= redefined connection coords of sub structure 1
% ni2= redefined internal coords of sub structure 2
% nc2= redefined connection coords of sub structure 2
%
FRF_Synth_data9
%
t0=clock;
flops(0);
count=0;
% extract the rows of phi relating to the coordinates of interest
phired1=phi1(ci1,:);
```

```
phired2=phi2(ci2,:);
for Comega=.1:.5:500
count=count+1;
%
%           Form Frequency Response Models for Each Substructure
%           ~~~~ ~~~~~~~~~ ~~~~~~~~ ~~~~~~ ~~~ ~~~~ ~~~~~~~~~~~~
for i=1:6
nomega1(i)=1/(omega1(i)^2-Comega^2);
end;
h1=phired1*diag(nomega1)*phired1';
for i=1:6
nomega2(i)=1/(omega2(i)^2-Comega^2);
end;
h2=phired2*diag(nomega2)*phired2';
%
a=size(i1);
b=size(c1);
c=size(i2);
d=size(c2);
%
aa=a(2);
bb=b(2);
cc=c(2);
dd=d(2);
%
%   Remember, we are trying to calculate the following:
%
%   hic* = hic - hic * M * inv( hccr ) * M' * hcc
%   hccr = M' * hcc * M
%
%  So we need to assemble [hic] and [hcc] using the
%  the coordinate sets we just defined.
%  These matrices contain the FRF data for both substructures
%  prior to coupling, i.e the pre-synthesis FRF data.
%
%
%                    Coordinate Partitioning
%                    ~~~~~~~~~~ ~~~~~~~~~~~~
%  Build up uncoupled FRF matrix and sub-partitions:
%
hic = [h1(ni1,nc1) zeros(aa,dd)];
%
hcc =  [h1(nc1,nc1) zeros(bb,dd);
        zeros(dd,bb)  h2(nc2,nc2)];
%
%  We can now perform the synthesis:
hccr=M' * hcc * M;
hicstar = hic - hic * M * inv( hccr ) * M' * hcc;
%
% look at the coordinate of interest
HH(count)=hicstar(1,3);
```

```
end;
etime(clock,t0),flops
Comega=.1:.5:500;
Freq=Comega/(2*pi);
plot(Freq,20*log(HH)),grid
xlabel(' Frequency Hz ')
ylabel('FRF at coordinate of interest dB')


%                    Called FRF_Synth_data9
%
% This is the data file for the synthesis program using modal
% representation.
%
% the following data will be provided by this file
%
% i1- internal coords sub structure 1
% i2- internal coords sub structure 2
% c1- connection coords sub structure 1
% c2- connection coords sub structure 2
%
% These are the coordinates of interest and are not the complete set.
% The connection coordinates are complete.
% The internal are the coordinates we wish to keep.
% This is used for defining the zeros partition
i1=[2];
c1=[4 5 6];
i2=[];
c2=[1 2 3];
%
% The coordinates of interst are internal and connection and the set
% is redefined. This is used for defining the reduced phi matrix
ci1=[2 4 5 6];
ci2=[1 2 3];
%
% the following is the mapping matrix
% the mapping matrix is not general and is
% case specific
%
M=[eye(3);
   -1*eye(3)];
%
% H1 and H2 are in the form of the coordinates kept for each structure.
% These reduced H matrices are (ci1xci1) and (ci2xci2). The internal
% and connection positions of h1 and h2 are redefined by there position
% in H1 and H2 (example internal coord 2 is now position 1 and
% connection coordinate 4 is position 2 etc)
%
% new coordinates
ni1=[1];
nc1=[2 3 4];
```

```
nc2=[1 2 3];


clear
clg
%                    FRF_Synth9 hcc reduced
%
% This program is used to determine the natural frequencies of
% the synthesized structure. We are interested in plotting the
% determenent of hcc reduced
%
%
% Load data from running unifineel program for each substructure.
% The K and M matrix for each structure is saved
load ex9a.mat     % k1 m1 is stored here
load ex9b.mat     % k2 m2 is stored here
% calculate the eigenvectors and eigenvalues for each structure
[lambda,phi]=fgmodes(k1,m1);
lambda1=lambda;
% calculate nat freq in rad/sec
omega1=sqrt(lambda1);
phi1=phi;
[lambda,phi]=fgmodes(k2,m2);
lambda2=lambda;
% calculate nat freq in rad/sec
omega2=sqrt(lambda2);
phi2=phi;
%
%  We need to create a single FRF matrix representing
%  both substructures in the form:

%      [hee] =  [ h(i,i)   h(i,c) ]
%               [ h(c,i)   h(c,c) ]
%
%  So we create arrays containing the DOF numbers of our original
%  models which correspond the the "c" and "i" coordinates for
%  each substructure.
%
% call the synthesis data file in now which contains the
% internal coordinates and connection coordinates for each sub
% structure.
% i1= internal coords of sub structure 1
% i2= internal coords of sub structure 2
% c1= connection coords of sub structure 1
% c2= connection coords of sub structure 2
% ci1= coordinates of interest of sub structure 1
% ci2= coordinates of interest of sub structure 2
% ni1= redefined internal coords of sub structure 1
% nc1= redefined connection coords of sub structure 1
% ni2= redefined internal coords of sub structure 2
% nc2= redefined connection coords of sub structure 2
```

```
%
FRF_Synth_data9
%
t0=clock;
flops(0);
count=0;
% extract the rows of phi relating to the coordinates of interest
phired1=phi1(ci1,:);
phired2=phi2(ci2,:);
for Comega=.1:.5:500
count=count+1;
%
%           Form Frequency Response Models for Each Substructure
%           ---- --------- -------- ------ --- ---- ------------
for i=1:6
nomega1(i)=1/(omega1(i)^2-Comega^2);
end;
h1=phired1*diag(nomega1)*phired1';
for i=1:6
nomega2(i)=1/(omega2(i)^2-Comega^2);
end;
h2=phired2*diag(nomega2)*phired2';
%
a=size(i1);
b=size(c1);
c=size(i2);
d=size(c2);
%
aa=a(2);
bb=b(2);
cc=c(2);
dd=d(2);
%
%   Remember, we are trying to calculate the following:
%
%   hccr = M' * hcc * M
%
%  So we need to assemble [hcc] using the
%  the coordinate sets we just defined.
%  These matrices contain the FRF data for both substructures
%  prior to coupling, i.e the pre-synthesis FRF data.
%
%
%                    Coordinate Partitioning
%                    ---------- ------------
%  Build up uncoupled FRF matrix and sub-partitions:
%
hcc =  [h1(nc1,nc1) zeros(bb,dd);
        zeros(dd,bb)  h2(nc2,nc2)];
%
%  We can now perform the synthesis:
```

```
hccr=M' * hcc * M;
% determine the nat freq of the synthesized structure
dhccr=det(hccr);
dh(count)=dhccr;
end;
etime(clock,t0),flops
Comega=.1:1:500;
Freq=Comega/(2*pi);
axis([0 80 -1e-12 1e-12])
plot(Freq,dh),grid
xlabel(' Frequency Hz ')
ylabel('Determinent of Hcc reduced')
```

APPENDIX I GENERAL MATLAB FUNCTIONS

```
function [c,s]=ftrig(t,numel)
% This function is used to calculate the values of
% cos and sin. The input to the function is the beam
% element angle
%
for i=1:numel
if t(1,i)<.02 & t(1,i)>6.25
c(1,i)=1;
s(1,i)=0;
elseif t(1,i)<1.58 & t(1,i)>1.56
c(1,i)=0;
s(1,i)=1;
elseif t(1,i)>3.11 & t(1,i)<3.17
c(1,i)=-1;
s(1,i)=0;
elseif t(1,i)>4.68 & t(1,i)<4.74
c(1,i)=0;
s(1,i)=-1;
else
c(1,i)=cos(t(1,i));
s(1,i)=sin(t(1,i));
end
end
```

```
function [kel,mel] = felement6(l,WTD,I,E,A,r,c,s)
%
% This function is generating the elemental mass and
% stiffness matrix. Input is element length, weight density,
% area moment of inertia, young modulus, radius of gyration
% and the angel of the element with respect to horizontal x
% axis. Counter clock is positive angle and clock is negative % angle.
%
% element mass matrix
%
grav = 386.4;
%
mel(1,1) = 140*c^2 + 156*s^2;
mel(1,2) = -16*c*s;
mel(1,3) = -22*l*s;
mel(1,4) = 70*c^2 + 54*s^2;
mel(1,5) = 16*c*s;
mel(1,6) = 13*l*s;
mel(2,1) = mel(1,2);
```

```
mel(2,2) = 140*s^2 + 156*c^2;
mel(2,3) = 22*l*c;
mel(2,4) = 16*c*s;
mel(2,5) = 70*s^2 + 54*c^2;
mel(2,6) = -13*l*c;
mel(3,1) = mel(1,3);
mel(3,2) = mel(2,3);
mel(3,3) = 4*l^2;
mel(3,4) = -13*l*s;
mel(3,5) = 13*l*c;
mel(3,6) = -3*l^2;
mel(4,1) = mel(1,4);
mel(4,2) = mel(2,4);
mel(4,3) = mel(3,4);
mel(4,4) = mel(1,1);
mel(4,5) = mel(2,1);
mel(4,6) = 22*l*s;
mel(5,1) = mel(1,5);
mel(5,2) = mel(2,5);
mel(5,3) = mel(3,5);
mel(5,4) = mel(4,5);
mel(5,5) = mel(2,2);
mel(5,6) = -22*l*c;
mel(6,1) = mel(1,6);
mel(6,2) = mel(2,6);
mel(6,3) = mel(3,6);
mel(6,4) = mel(4,6);
mel(6,5) = mel(5,6);
mel(6,6) = mel(3,3);
%
% now calculate gamma.
% gamma is mass density per unit length and grav is the
% gravitational constant in in/sec^2.
%
gamma = WTD*A/grav;
%
% now apply the mass constant to the elemental matrix
%
mel = mel*(gamma*l/420);
%
% now make the element stiffness matrix
%
kel(1,1) = (1/r)^2*c^2 + 12*s^2;
kel(1,2) = (1/r)^2*c*s - 12*c*s;
kel(1,3) = -6*l*s;
kel(1,4) = -(1/r)^2*c^2 - 12*s^2;
kel(1,5) = -(1/r)^2*c*s + 12*c*s;
kel(1,6) = -6*l*s;
kel(2,1) = kel(1,2);
kel(2,2) = (1/r)^2*s^2 + 12*c^2;
kel(2,3) = 6*l*c;
```

```
kel(2,4) = -(1/r^2*c*s + 12*c*s;
kel(2,5) = -(1/r^2*s^2 - 12*c^2;
kel(2,6) = 6*l*c;
kel(3,1) = kel(1,3);
kel(3,2) = kel(2,3);
kel(3,3) = 4*l^2;
kel(3,4) = 6*l*s;
kel(3,5) = -6*l*c;
kel(3,6) = 2*l^2;
kel(4,1) = kel(1,4);
kel(4,2) = kel(2,4);
kel(4,3) = kel(3,4);
kel(4,4) = kel(1,1);
kel(4,5) = kel(1,2);
kel(4,6) = kel(4,3);
kel(5,1) = kel(1,5);
kel(5,2) = kel(2,5);
kel(5,3) = kel(3,5);
kel(5,4) = kel(4,5);
kel(5,5) = kel(2,2);
kel(5,6) = kel(3,5);
kel(6,1) = kel(1,6);
kel(6,2) = kel(2,6);
kel(6,3) = kel(3,6);
kel(6,4) = kel(4,6);
kel(6,5) = kel(5,6);
kel(6,6) = kel(3,3);
%
% now apply the stiffness constant to the elemental matrix
%
kel=kel*(E*I/(l^3));


function[lambda,phi]=fgmodes(kg,mg)
%
% This function calculates the natural frequencies and
% the mode shapes. Theses are the eigenvalues and
% eigenvectors
%
a=length(mg);
[V,D]=eig(mg\kg);
[omga,index]=sort(diag(D));
lambda=zeros(a,a);
for i=1:a;
lambda(i,i)=omga(i);
end;
for i=1:a;
phitemp(:,i)=V(:,index(i));
end;
lambda=diag(lambda);
[phi,orth]=fgmassnorm(phitemp,mg);
```

```
function[phinorm,orth]=fgmassnorm(phi,mass
%
% This function mass normalizes the eigenvectors
%
a=size(phi);
nmodes=a(1,2);
phinorm=zeros(phi);
for ii=1:nmodes;
modalmass(ii)=phi(:,ii)'*mass*phi(:,ii);
if modalmass(ii)~=0
phinorm(:,ii)=(1/sqrt(modalmass(ii)))*phi(:,ii);
else
phinorm(:,ii)=phi(:,ii);
end;
end;
% now check orthoganality
orth=phinorm'*mass*phinorm*100;


function[kdsprg]=fsprngdamp(k,Comega,B);
%
% This function will generate a spring and damper system k
% matrix of size [6 x 6] which correlates with the 3 dof of
% the beam element kx, ky, k(theta), cx, cy, c(theta).
% Remember that for a spring of stiffness k and a damper
% with damping ßk, the matrix looks like
%       [kx + jΩßk   -kx - jΩßk
%       -kx - jΩßk    kx + jΩßk].
% k has the units of lbs/in
%
kdsprg = zeros(6);
kdsprg(1,1) = k + j*Comega*B*k;
kdsprg(1,4) = -k - j*Comega*B*k;
kdsprg(2,2) = k + j*Comega*B*k;
kdsprg(2,5) = -k - j*Comega*B*k;
kdsprg(3,3) = k + j*Comega*B*k;
kdsprg(3,6) = -k - j*Comega*B*k;
kdsprg(4,1) = -k - j*Comega*B*k;
kdsprg(4,4) = k + j*Comega*B*k;
kdsprg(5,2) = -k - j*Comega*B*k;
kdsprg(5,5) = k + j*Comega*B*k;
kdsprg(6,3) = -k - j*Comega*B*k;
kdsprg(6,6) = k + j*Comega*B*k;

function[kdsprg]=fsprngdampC(k,Comega,q);
%
% This function will generate a spring and damper system k
% matrix of size [6 x 6] which correlates with the 3 dof of
% the beam element kx, ky, k(theta), cx, cy, c(theta).
% Remember that for a spring of stiffness k and a damper with
% damping (Co e^-qΩ), the matrix looks like
```

```
%       [kx + jΩc(Ω)   -kx - jΩc(Ω)
%       -kx - jΩc(Ω)    kx + jΩc(Ω) ]
% k has the units of lbs/in
%
fe=exp(-q*Comega);
kdsprg = zeros(6);
kdsprg(1,1) = k + j*Comega*k*fe;
kdsprg(1,4) = -k - j*Comega*k*fe;
kdsprg(2,2) = k + j*Comega*k*fe;
kdsprg(2,5) = -k - j*Comega*k*fe;
kdsprg(3,3) = k + j*Comega*k*fe;
kdsprg(3,6) = -k - j*Comega*k*fe;
kdsprg(4,1) = -k - j*Comega*k*fe;
kdsprg(4,4) = k + j*Comega*k*fe;
kdsprg(5,2) = -k - j*Comega*k*fe;
kdsprg(5,5) = k + j*Comega*k*fe;
kdsprg(6,3) = -k - j*Comega*k*fe;
kdsprg(6,6) = k + j*Comega*k*fe;


function[Trmatrix]=ftrans(c,s,beam)
%
% This function generates the transformation matrix
% used in the finite element program
%
Trmatrix=zeros(6,6);
Trmatrix(1,1)=c(beam);
Trmatrix(1,2)=s(beam);
Trmatrix(2,1)=-s(beam);
Trmatrix(2,2)=c(beam);
Trmatrix(3,3)=1;
Trmatrix(4,4)=c(beam);
Trmatrix(4,5)=s(beam);
Trmatrix(5,4)=-s(beam);
Trmatrix(5,5)=c(beam);
Trmatrix(6,6)=1;


function[Trmatrix]=frftrans(ccc,ss)
%
% This function generates the transformation matrix
% used in the synthesis program
%
Trmatrix=zeros(6,6);
Trmatrix(1,1)=ccc;
Trmatrix(1,2)=ss;
Trmatrix(2,1)=-ss;
Trmatrix(2,2)=ccc;
Trmatrix(3,3)=1;
Trmatrix(4,4)=ccc;
Trmatrix(4,5)=ss;
```

```
Trmatrix(5,4)=-ss;
Trmatrix(5,5)=ccc;
Trmatrix(6,6)=1;
```

LIST OF REFERENCES

1. Gordis, J.H..Bielawa, R.L., Flannelly, W.G., "A General Theory for Frequency Domain Structural Synthesis," *Journal of Sound and Vibration*, Vol. 150 No. 1. pp. 139-158, 1989.

2. Gordis, J. H., Flannelly, W.G. *Analysis of Stress Due to Fastener Tolerance in Assembled Components*. Proceedings of the 34th AIAA/ASME/ASCE/AHS/ACS Structures, Structural Dynamics, and Materials Conference. La Jolla, CA., 1993.

3. Gordis, J. H. *Structural Synthesis in the Frequency Domain: A General Formulation*. To appear, Proceedings of the 12th International Modal Analysis Conference, Honolulu, HI., 1993.

4. James, M.L., Smith, G.M., Wolford, J.C., Whaley, P.W., *Vibration of Mechanical and Structural Systems*, Harper and Row, Publishers, Inc., New York, NY. pp. 482-572, 1989.

INITIAL DISTRIBUTION LIST

		No. Copies
1.	Defense Technical Information Center Cameron Station Alexandria, Virginia 22304-6145	2
2.	Library, Code 52 Naval Postgraduate School Monterey, California 93943-5002	2
3.	Professor J. H. Gordis, Code ME/Go Department of Mechanical Engineering Naval Postgraduate School Monterey, California 93943	3
4.	Professor A. Healey, Code ME/He Department of Mechanical Engineering Naval Postgraduate School Monterey, California 93943	1
5.	Department Chairman, Code ME Department of Mechanical Engineering Naval Postgraduate School Monterey, California 93943	1
6.	Naval Engineering Curricular Office (Code 34) Naval Postgraduate School Monterey, California 93943	1
7.	LT. Ronald E. Cook 1090 Derby Street Casper, Wyoming 82609	1