Integrating Experimental and Finite Element Method Modal Analysis

by

Patrick Ryan Turner Engineering Manager of MCAE

Schlumberger Technologies, CAD/ CAM Division,
Ann Arbor, Michigan, USA 48106

May 17, 1989

 Presented at:

Spring Users Group Conference

"Engulfed by CAD/CAM"

Tampa, Florida

May 21-25, 1989

Abstract

This paper will first give a basic overview of modal analysis and its use. Next, the merits of integrating experimental and analytical modal analysis will be discussed. The advantages of modal integration provided to the engineer go beyond the ability to compare theoretical and experimental test results. Data exchange and new features made possible by this integration make the CAE design process more efficient.

GRAFEM has been enhanced to communicate with a modal industry standard file format. The implementation will be discussed with specific reference to the Schlumberger Instruments 1202 Structural Analyzer. This unit is a unique self contained experimental modal analysis and data acquisition system.

1 INTRODUCTION

Since engineers started analyzing designs on the computer, the question has always been asked, "How do the results correlate with the experiments?" Before there was the ability to analyze designs on the computer, there was always the cut-and-try method of design. This trial-and-error process required the manufacture of many costly prototypes and rigorous experimental testing. As mechanical design markets have become more competitive, ways to short-cut this costly and time consuming process have been required. So, direct integration between traditionally analytical (computer based) analysis and experimental (physical measurement based) analysis has been part of a logical progression. In particular, the integration of experimental and FEM modal analysis gives the engineer a new level of decision making productivity and most of all confidence. Let's examine this statement.

This logical progression is based on the premise that computer based modeling and analysis of designs is faster than building and testing many prototypes. Another assumption is that the computer model reflects the true characteristics of the real part. Actually neither of these statements can be considered a given. In fact, the only thing we can count on is the flexibility of the computer which provides the *potential* of these statements to be true. In this light, the integration of experimental and FEM modal analysis will be investigated as it is implemented in the GRAFEM finite element modeling system. From this discussion we will try and conclude what added capability is provided to the engineer/analyst by this type of integration.

To facilitate this discussion a brief review of modal analysis and its uses will be given from the layman's perspective. Then a description of the GRAFEM interface to modal analysis will set the stage for a discussion of the merits of experimental and FIEM modal analysis integration. Finally, a description of the Schlumberger Instruments 1202 Structural Analyzer will be provided to show how Schlumberger Technologies has provided a unique mechanical design environment through the cooperative efforts of the CAD/CAM and Instruments Divisions.

2 LAYMANS OVERVIEW OF MODAL ANALYSIS

Before starting, two terms must be defined that will be used throughout this document.

Analytical refers to any procedure or process that obtains information based on theory or mathematics. This procedure gathers little or no measured information from the physical system or part being analyzed. Finite Element Analysis, hand calculations, computer simulations, etc. are analytical engineering tools.

Experimental refers to any information that is obtained based on an experiment or measurement on the physical system or part. A volt meter, oscilloscope, accelerometer, modal analyzer, or strain gauges are a few tools used in experimental analysis to obtain information about a part or system.

Every part and structure has many so-called resonance frequencies determined uniquely by the part's shape and material. When excited by the proper frequency, the part resonates and vibrates vigorously. The frequency and the shape of vibration are unique to a particular part.

This behavior is seen in everyday life. Does your car rattle? If it does, the pitch of the rattle should always be the same regardless of the speed of the car. However, the intensity may vary depending on the speed. Also, if you've ever held a ruler or yard stick over the edge of a table a good distance and tapped the other end, you have seen a resonant frequency. The speed at which the ruler vibrates is it's resonant frequency; the more that hangs off the table, the lower the frequency. So, changing how a part is used or mounted can change it's frequency.

A musical instrument is a rare example of employing resonance to our benefit, but in general, resonance vibration is something to be avoided or it may damage the part or the structure. For example, if a part vibrates too violently or for too long it may crack and break. This is seen too often in machine designs. A part mounted to a machine may vibrate during the machines operation, say, a muffler guard on your lawn mower. Eventually small cracks will appear near the mount connection. As the vibration continues, the crack will grow until the part breaks off completely.

Each part has several distinctive resonant frequencies that it will vibrate at. Counting from the lowest, they are called the first, second, third, etc. natural or resonant frequency. Frequency is usually expressed in terms of cycles per second, which is a unit of measure called hertz (Hz). When a resonance is related to rotating machinery, it is specifically called the "critical speed" and expressed in terms of revolutions per minute. So critical speed and resonance frequency are actually the same phenomenon expressed in different units.

For each resonant frequency, the part vibrates in a particular unique shape, called a mode shape. If there is no damping (a way to dissipate the vibration), the smallest exciting force will result in unlimited vibration amplitude. However, thanks to the material damping (friction between microscopic particles of the material), the vibration amplitude is limited to a certain level and the smaller the exciting force, the smaller the resulting vibration.

The kind of questions that need to be answered when vibration problems exist are:

- what are the critical speeds or resonant frequencies of a structure,
- what is the vibrating mode shape corresponding to these frequencies, and
- how much stress or deformation will actually occur.

Equipped with the answers to these questions, the parts can be modified to avoid the resonance, withstand the stress, or reduce the deformation.

Modal analysis is used to find the answers to these questions. The required modal analysis can be performed experimentally or analytically. Finite element analysis is an analytical technique which predicts the behavior of the part through linear dynamic (eigenvalue) analysis. The experimental technique measures the vibration of an actual part using devices called accelerometers[1], shakers

[1] These devices will be discussed in a little more detail in the next section.

(or impact hammers), signal processing equipment, etc. Then, the vibrational characteristics are extracted from the data using a modal analyzer.

Most parts are designed to have the first resonant frequency far above the operating frequencies so that the part will never resonate. Winchester disk drives, VCR's, audio turn-tables, etc. all are designed to be stiff enough so that the first natural frequency is above the operating speed of the internal moving components. But in some cases, it is impractical to design the part that way because the part becomes too heavy and big. In this case, the operating speed is set between two of the structures resonant frequencies. This is the case for most high speed rotating machines. Instantaneous high vibration may be noticed when a jet engine is started and goes through the first critical speed. Or, a car engine may shake violently as it comes to a stop or is started. It is very common to let machines go through a resonant frequency, but most designs force this to happen very quickly.

if the shape of the part is simple, the resonant frequency can be determined from hand calculations and avoided while the part is being designed. However, most parts are not that simple. So, it's difficult, if not impossible, to find these resonant frequencies before the part is built. Unless, a tool like finite element analysis can be applied.

Finite element analysis can be very useful in analyzing a part's vibration BEFORE it is manufactured. However, FEA involves many assumptions and simplifications which are sometime not acceptable in precise design problems. If damping or nonlinear boundary conditions are an important design consideration, then standard FEA is difficult to apply with any accuracy. However, in most cases, FEA can be successfully used to suggest design trends and improvements.

Once FEA has helped guide the conceptual design phase, the finished parts can be analyzed using Experimental Modal Analysis (EMA). EMA can determine the true characteristics of the part, assuming the analysis is performed correctly. The results can be checked against the theoretical model to verify the assumptions and simplifications. Once verified, the theoretical model can be used with more confidence when the part goes to the detailed design iteration phase.

3 HOW A MODAL ANALYZER IS USED

Experimental Modal Analysis is performed by using an analyzer, acceleration sensors (called accelerometers), a vibration input device (a shaker or impact hammer may be used), force transducers, and data collection equipment (also called signal processing or data acquisition equipment). Very often an "impact hammer" equipped with a force sensor (to measure the input) is used as the source of vibration. The part is placed in a test fixture (how it is attached to the fixture will have a definite impact on the results), accelerometers (vibration sensor) are attached at certain strategic locations and the part is hit by the hammer or shaken by a shaker (a shaker is like a very large audio speaker coil used to shake/vibrate a part at a particular frequency or frequency range).

At the heart of any modal testing analysis is a data acquisition system that uses a technology called FFT (Fast Fourier Transforms). FFT technology coverts vibration measurements from a time based signal (mag/time) to a frequency based signal (mag/frequency); and does this in real time. In this way the vibration is decomposed in terms of what frequencies make up the total vibration. The raw data in this form is called a "FRF" or "Frequency Response Function".

After the data is collected the modal analyzer calculates the natural (resonant) frequencies, damping coefficients, and a parameter called the residue from each measurement (FRF) using a procedure called curve fitting. Finally, once all of these factors are known, the system reconstructs the vibration mode shape and it can be displayed on the screen. This is done by an analysis of the relationship between the input to each output point. This analysis provides a total vibration "picture", relating all the output points to each other with proper motion timing (phasing) between them. Recall in this case, the hammer impact or shaker is the input causing the structure to vibrate. The vibration is measured by the accelerometer (vibration sensor) as output.

The location of the sensors and the vibration input have a vital impact on the test results. For example, not placing sensors at locations with local motion (vibration isolated to a small area of the structure) can cause significant errors. So, the data from ill placed sensors does not have much value, therefore, careful pre-test study is a very good idea.

As with any computer analysis technology, modal analysis is most effective when used as early as possible in the design cycle. In reality however, EMA is more often used to solve acute vibration problems on systems already in service. An engine installed on the 20th floor which operates a freight elevator might vibrate and shake the whole building. A little sensor attached with a bracket to a engine which resonates and falls off, etc. It may be impractical to rebuild the building, the engine, or the sensor. However, some action must be taken to solve the problem. Modal analysis is very useful in telling the engineer what action to take. In most cases, the options are limited as a result of design or application constraints. These options usually include changes to the structures mass, stiffness, or damping characteristics. By performing a modal analysis the problem can be characterized and likely solutions identified. With the elevator, if the vibration of the motor cannot be changed, then the motor must be isolated from the building using some kind of isolating mounts. Using EMA the mount can be specified so it would prohibit the vibration from being transmitted to the building. A problem can occur if a resonant frequency coincides with a machines operating speed, like the sensor bracket combination. Then, changing the stiffness of the bracket is likely the most effective solution (assuming significant changes to the mass is undesirable). Another type of isolation problem exists if the vibration is *forced* by the machines operation (indicated by the fact the resonant frequencies do not occur at the offending frequency). Again, isolating the vibration by some how changing the system damping characteristics (rubber bumpers, soft mounts, etc.) is likely the most effective solution.

In summary, equipped with the data from the modal analyzer and a computer model, the effect of proposed actions (such as adding a stiffener or damper) can be quickly and efficiently analyzed. So, it's possible to come up with a solution in a short period of time.

4 Interfacing GRAFEM to Experimental Modal Analysis

The "neutral file", also called the universal file, is the method used to implement the integration of experimental modal analysis and finite element method modal analysis within GRAFEM. The neutral file is widely accepted as the standard for transferring modal information. By using the neutral file, GRAFEM is not restricted to communication to any particular modal analysis system. Therefore, GRAFEM allows the processing of a neutral file created by or for a modal analyzer such as the Schlumberger Instruments 1202 Modal Analyzer. Through this connection, modal comparison between theoretical and experimental analysis can be made. The advantages of this type of connection will be explored in the next section. The neutral file is broken up into data set types that are used to communicate various types of data within engineering systems.

The neutral data set types utilized in GRAFEM include set 15 (grid points), set 55 (modal vectors), set 58 (frequency response functions), and set 82 (trace links). Each of these data set types are defined in the appendix of the GRAFEM Command Reference Manual.

The GRAFEM finite element model can be created with no consideration to the desire to transfer data to the modal analyzer. This allows the analyst to carry on standard structural analysis and only give consideration to the modal data transfer when needed. This implies the finite element model can be created using any standard element type. Once the model has been created and the analyst wants to transfer modal results to the analyzer, the finite element counterpart to modal analysis must be executed. This analysis counterpart is referred to linear dynamic analysis (or elgenvalue analysis) in finite element. This analysis type creates the natural frequencies and modal vectors (or mode shapes) needed for transfer to the analyzer.

4.1 Writing an Experimental Modal Analysis Input Data File

Once the model has been created and the finite element results obtained, creation of experimental modal analysis neutral file may begin. The system prompts for the neutral file name to be created and then gives two options pertaining to how trace lines can be created. The trace lines can be created with the BYNODE option by selecting nodes on the existing finite element model or with the BYELEMENT option by selecting existing beam elements that will be converted to the trace lines themselves.

The BYNODE option indicates that trace lines are to be defined by picking nodes one at a time. Trace lines will be drawn in red as they are being defined. Many separate trace lines may be defined during a neutral file transfer. A given trace line is terminated by entering DONE, at which time the current trace line color will change to magenta. Then, subsequent trace lines may be created by just continuing to select nodes. Termination of the trace line definition process occurs when an extra DONE is entered following the latest trace line definition. Any trace line can be “un-created” a node at a time during the creation process by selecting BACKUP. The trace line will be deleted on a most recently selected node basis.

If node numbers greater than 300 are detected in the node list, all nodes output will be convert sequentially from one maintaining the same relative order. This is done because most modal analyzers have a limit on node number (response point) values.

The BYELEMENT option indicates that trace links will be defined by linear beam elements already existing in the finite element model. An option will be issued that allows the selection of elements, only selected beam elements will be used. Also, only the nodes referenced by the selected beams will be output. If node numbers greater than 300 are detected in the node list, all output nodes will be converted sequentially from one maintaining the same relative order.

Once all trace lines have been defined, the system will issue the message:

```
Data input complete. . . . starting MODAL ANALYZER neutral file write
```

4.2 Reading Experimental Modal Analysis Results

Several strategies are provided for transferring modal analysis results from the modal analyzer. The neutral file data types supported during the "read" operation are the same as those supported during the "write" operation. After prompting for the neutral file name to read the system allows the modal model to be reoriented so that it aligns with the finite element model. This is very valuable if the modal model is created using a different coordinate system reference than what was used while creating the finite element model. Or, if there is slight differences in the geometric scale of the modal model compared with the finite element model.

If model reorientation is desired, the user is prompted for three common nodes (or measurement points) between the modal test model and the FEM model. This is done by prompting the user for a finite element node and the corresponding modal test response point. Note: the 3 nodes must form a plane. Once these three pairs of points are entered, the system can create the proper transformation and scaling matrix. This matrix is then used to reorient and scale the modal model to align with the finite element model.

This reorientation option may be overridden and the modal analysis model transferred using the coordinates that are contained within the neutral file. The next choice given whether to MERGE or OVERLAY the modal model onto the finite element model.

The MERGE option substitutes GRAFEM nodes for the analyzer nodes to which they are closest. If reorientation of the incoming model was previously specified, the MERGE will be conducted following the realignment. NOTE: Nodes must be merged if results currently exist in the database in order to maintain the integrity of the currently existing result sets.

The OVERLAY option adds new nodes to the database. This option is only offered if the current database does not contain any result sets.

Finally, the system will issue the message:

```
Data input complete....starting MODAL ANALYZER neutral file read
```

5 MERITS DERIVED FROM MODAL INTEGRATION

5.1 Use of the Neutral or Universal File

The integration of these two modal analysis technologies is implemented using an industry-standard file format. This file format is called the "Neutral" or "Universal" file format and is used to transfer a diverse set of geometric and analytical data. When manufacturers of modal equipment are interested in transferring data to outside systems this is usually the format of choice. This allows GRAFEM to supply data for, and read data from, a broad range of systems in addition to the Schlumberger Instruments 1202 Structural Analyzer.

5.2 Single source of supply

Schlumberger Technologies is the first company to design and manufacture both of these MCAE tools; the FEM software and the modal analyzer. Being able to obtain both the experimental and FEM modal analysis capabilities from a single source can be a significant advantage. A single source of supply means no communication problems between the manufacturer of each component, better integration, and eliminates finger pointing between the suppliers of different components.

5.3 Integration Allows the Use of Only One Geometric Model

With the increased use of computers for design and analysis this integration removes one more area of duplicated effort. Before, when an Experimental Modal Analysis was desired, much data needed to be manually entered into the analyzer to prepare for the test (even if the data already existed in a CAD data base). This included physically laying out the modal measurement point locations on the structure to be tested, then measuring the location of these points from some reference location. Finally, these measured coordinates were manually entered into the analyzer. This was necessary to define the geometry of the part in the analyzer. In addition, deciding on the location of these points requires some experience and expertise. Determining the most effective measurement and excitation locations can be greatly aided by modal integration. This will be discussed more in the next section.

Now, the same model created by the CAD system and used for finite element analysis and manufacturing can be moved into the modal analyzer to define the object. Using Editor and GRAFEM functions will define the test object more quickly and accurately. The Editor and GRAFEM are designed for geometry and analysis model creation and only basic capabilities are provided for this in a modal analyzer. Therefore, using these systems means no more error prone repetitive work is necessary where the analyst must retype part or measurement point coordinates. Once the geometry is transferred to the analyzer, the analyst only has to visually match the response locations of the real structure to node locations on the geometric model. This can dramatically reduce the amount of measuring needed to place measurement points on the real structure.

5.4 The FEA Capability Can Improve and Supplement Modal Testing

If a company is inclined to use CAD and CAE then it is very likely that some analysis will have been done on a part before a prototype is made and therefore, before the experimental modal analysis is performed. If not, then at least a CAD model will exist. In this case, before conducting an experimental modal analysis, preliminary theoretical analysis can be performed to determine the most effective test setup. In fact, many modal testing experts recommend this procedure regardless.

A linear dynamic (eigenvalue) analysis can be performed on the structure. This will result in a set of natural frequencies and mode shapes (modal vectors). The mode shapes will show the locations where motion amplitude is the greatest for all modes. Placement of measuring devices should coincide with theses locations on the structure; these are locations that are likely to vibrate the most. This will assure that a strong signal will be measured by the analyzer in relation to unimportant instrumentation static (noise) for a given mode. Also, the location where the input motion (excitation) should be applied can be determined. This motion is used to excite the vibration of the physical part during the experimental test. Again, the input motion should be located on the structure where vibration levels are greatest. If the input motion is on the structure where no response is expected for a particular mode then, in reciprocal fashion, no vibration will be excited for that mode. So, it is important for the engineer to understand where to place these devices. If these important locations are not determined through FEA analysis or through experience, then usually more than the necessary number of points will be used in the modal test to insure the entire response is measured. So, the use of FEA to determine the best test setup will result in a more efficient test procedure and reduce cost and time required for the test considerably.

5.5 Enhanced Capabilities Possible with Modal Integration

Modal integration can provide some tools to the engineer that were not previously available, or at least not convenient. These tools provide additional information about the design or integrity of analysis results. FEM (finite element method) modal analysis results can be imported into the experimental modal analyzer for additional analysis. When the experimental and analytical data are combined, the engineer can gain much more information by performing Modal Assurance Criterion (MAC) calculations, FRF synthesis, or analytical structural modifications. These techniques are described below.

5.5.1 Modal Assurance Criterion (MAC)

Up until now, doing correlation between experimental and analytical results has been a relatively subjective or tedious process. Now, by transferring FEA results into the experimental modal analyzer, MAC (Modal Assurance Criterion) calculations can be performed. MAC is a quantitative comparison between two mode shapes. In this case, the FEA and experimental mode shape for the same mode are used in the calculation. For a given mode, the MAC value should be one for perfect correlation. This calculation compares how closely the FEA model matches the real part (or at least how well it matches the experimental modal analysis results). Then, if the modes match well, the FEM model can be used to investigate design changes more cost effectively. Modification of the FEA model is faster that making hardware changes to the structure and re-running experimental tests. If the modes don’t match well, other methods of comparing modal analysis results can be used. Visual mode shape comparison will sometimes indicate one of the models (experimental or FEM) lacks detail. This happens when one model is more course than the other and data is missed. For modal testing, only the response measured is use to create results. Therefore, if not enough points are used to pick up some local motion then, mode shapes will not be defined well. Similarly, for FEM modal analysis, there is a concept that implies that enough nodes (degrees-of-freedom) must be used to “capture” the response.

5.5.2 FRF (Frequency Response Function) Synthesis

Another method used to verify the results of a modal test is to use a procedure called FRF Synthesis. FRFs or Frequency Response Functions are the measured raw data (the measured relation between the input and output, as discussed in previous sections) used to construct modal analysis results. These measurements can be individually reconstructed (synthesized) from the modal results. Then, the synthesized FRF can be compared to the actual measured FRF to get an indication of the quality of the modal analysis.

This same procedure can be used to compare an analytical and experimental modal analysis. FEM modal analysis is much different from modal testing. The FEM procedure does not use FRFs at all. None-the-less, the mode shapes and resonant frequencies (modal results) from the FEM analysis can be used to synthesize FRFs. This FEM based synthesized FRF can be compared to actual measured FRF from a modal test of the same part. If the FEM model portrays the same modal characteristics as the actual part the FRFs should be close to the same in appearance.

5.5.3 Substructured Analysis and Structural Modification

Substructured analysis is a procedure used primarily to break up large analyses for computational efficiency. For example, if changes to a system are restricted to a small area, then subsequent analyses or tests need not reconsider the unchanged portion. This technique has been applied in both FEM and modal testing. However, with modal integration, the concept of substructured analysis provides capabilities not previously available.

This new capability starts by transferring FEM modal results to the Schlumberger Instruments 1202 Structural Analyzer. Because of modal integration the FEM theoretical analysis results of one part can be combined with the experimental results of a mating part. Next, the two modal models are combined at the modal coefficient level. This procedure gives results similar to those created when the two parts are analyzed as a whole. The implication is that the modal characteristics of the separate parts interact creating results that can be somewhat different than a simple superimposed combination might create. This substructured analysis will greatly enhance the cost effectiveness and efficiency of the design process.

For example, substructured analysis can be applied when one part of an assembly is going to be redesigned. In this case, there is no need to build a costly prototype of the part to do the experimental test. The new part of the assembly can be analyzed with finite elements on the computer without building a prototype. The results of the FEM modal analysis can be transferred to the 1202. Then, if not completed ahead of time, the existing mating part(s) can be tested experimentally with the 1202 analyzer. Finally, the two results can be combined in the analyzer to give the results for the whole system. This was impossible before these technologies were integrated!

To add even more capability, using the structural modification section of the 1202, changes to the assembly can be evaluated in the analyzer before any additional FEM model or hardware changes are made. This is particularly useful when the modal characteristics of the two structures combine in some nonlinear (non-intuitive) fashion. Thereby confusing how one of the components might be changed to fix a problem.

6 Schlumberger Instruments 1202 Structural Analyzer Description

The Schlumberger Instruments 1202 Structural Analyzer[2] is a complete four-channel structural analysis system integrated into a single, portable unit. It includes software for FFT signal processing, modal analysis, structural modification, and forced response simulation. It has internal fixed and floppy disk drives, IEEE-488 and RS232 interfaces, and a signal generator. Fast graphics on

[2] The information in this section was extracted from Schlumberger Instruments literature entitled *1202 Structural Analyzer* [1].

the large built-in display features hidden line animation of mode shapes, strain contour maps, and flexible stack plots.

Combined with the Solartron 1250 series Frequency Response Analyzers it can be expanded to provide up to 36 parallel sine channels for fast, accurate results in large scale tests. This ensures that the system can grow with the requirements of the engineer.

6.1 1202 SIGNAL PROCESSING

The signal processing section of the 1202 features four channel 40kHz FFT analysis with zoom and integrated signal generation.

It can be operated using special-purpose keys or through self-explanatory menus and soft keys. All operations can be programmed for repeated tests. Because the system can define multiple frequency ranges for simultaneous analysis, up to one thousand lines of resolution are available.

The wide range of time and frequency domain functions includes input time history, transfer function, transmissibility, impulse response, coherence, coherent output power, and multiple input transfer function.

All functions can be shown on single, dual, or quad displays, or can be added to a stack display with variable skew and direction. Sequential frequency response acquisition and storage for a group of measurement locations can be simplified using a special menu page, and automated using the learn facility.

The system provides a big buffer facility allowing 1.5 million input samples to be collected for later analysis. It can be used to speed analysis of tape recorded signals, and can give an effective real time rate up to 40kHz, the full frequency range of the instrument.

6.2 1202 MODAL ANALYSIS

The 1202 Structural Analyzer includes and extensive tool kit of modal analysis features making it applicable in all testing situations. Its advanced geometry entry and manipulation system simplifies the interpretation of complex structures, with mode shape displays enhanced by fast animation, hidden line suppression, surface contours, and cross-sectioning.

Multi degrees-of-freedom curve fitting can be as interactive or automated as the test case requires, with automatic sorting of fitted results for global values. Interpolation equations can be set up for points with no measured data to clarify displays. Structural modification and forced response simulation facilities, often regarded as accessories, are included as necessary parts of the system. Features like modification sensitivity ranking and subsystem analysis may be essential to solve the vibration problem.

Modal analysis is closely linked with the signal processing system. It is operated in the same way, using special keys or menus, but also has a sequence of data tables which guide the user through the analysis. All signal processing features except the big buffer continue to be available any time.

All modal project data can be stored on fixed or floppy disks, with the option of automatic backup or reminder at preset time intervals. The system can be programmed either to automate an analysis sequence or to combine signal processing and modal analysis operations.

7 CONCLUSION

This “modal integration” provides *objective* comparison between test and FEM results using MAC and FRF Synthesis. However, the advantages provided to the engineer go beyond comparison of theoretical and experimental results. Providing substructured analysis and structural modification between physical and conceptual designs allow a new level of design process efficiency.

GRAFEM has been enhanced to communicate with a modal industry standard file format. Providing convenient options for how the modal data is transferred including: modal model reorientation, modal model merge or overlay, etc.

8 ACKNOWLEDGEMENTS

Toichi Aoyagi, Jugma Bora, and Mitchell Walsh of the CAD/CAM division contributed to several internal documents on which several sections of this paper are based. Their efforts have greatly added to this paper.

9 BIBLIOGRAPHY

1. *1202 Structural Analyzer*, Schlumberger Instruments 1987, SI/DYN/040/Issue 2/February 1988/FDA2186.