The past and present roles of computer-aided engineering in DASD design

by T. Kan
D. B. Lawson
O. J. Ruiz
C. F. Sermon

At the heart of today's computer-aided engineering (CAE) revolution is finite element modeling (FEM). This paper presents a brief history of how FEM simulations interact with and have significant impact on the design process of storage devices. The discussion is limited to structural static and dynamic effects on head/disk assemblies (HDAs) and components. FEM is integral to the design process; it is primarily a predictive/diagnostic design tool that provides engineers with detailed information on the performance of a design. FEM is most effective during the concept phase, where it can sort out many performance issues before the design parameters are constrained. Also, FEM can help to optimize critical structures within the system. As a diagnostic tool, FEM supplements testing by predicting in advance the properties and behavior of the device. A three-piece suspension design is presented as an example of how FEM and design work in harmony. An FEM of the entire structure was built to verify design and to fine-tune dimensions. Areas that required reinforcement and frequencies that seemed too low were identified, and the structure was modified. This process was repeated several times until the design satisfied the requirements. In addition to the suspension design example, a thermal deformation problem with a 3.5-in. actuator comb assembly is discussed.

Introduction

There are several categories of computer simulation, of which FEM is one that is particularly well suited to solid mechanics. In the application of FEM to solid mechanics, the analyst breaks the structure up into many small pieces (hence the name *finite element*), with each piece acting like a small linear spring. The overall stiffness of the structure can be determined by summing the effects of the many small springs. Boundary conditions are applied, and the deflection is estimated. This simple concept can have many applications, e.g., statics, thermal deformation, normal modes, frequency, and time domain response. DASD is therefore a field that is rich in FEM

0018-8646/96/$5.00 © 1996 IBM

applications, because all of those attributes must be analyzed in the process of DASD design.

The effectiveness of an FEM code in enhancing the design process is directly related to its speed, accuracy, and versatility. Today almost all commercial codes allow fast pre- and post-processing, sufficient accuracy, and a wide variety of boundary conditions and solutions.

Many problems affect modeling productivity, such as the absence of a robust pre/post-processor. CAEDS® was the first available tool, at IBM in San Jose, to provide interactive/menu-driven pre/post-processor capabilities. At San Jose, several FEM tools are available:

- CAEDS.
- ANSYS®.
- MSC/NASTRAN.
- NASTRAN® preprocessor.
- Various utility tools.

In contrast to the tools available in most industries, this menu represents a rich assortment of software tools that can be applied to structural mechanics problems.

The philosophy we follow is to use modeling early in the design process to provide trends and guidance. The model should be as simple as possible consistent with requirements. This approach takes less time to complete and is more accommodating of the model modifications that often follow.

Brief history

In the past, simulation was used in many applications, e.g., magnetics, air bearings, manufacturing yield, electronic circuits, thermal, fluids, track misregistration (TMR), structural mechanics, and servos. For HDA development at San Jose, many significant efforts helped to establish finite element (FE) modeling as part of the development process.

Notes from 1973[1] on using NASTRAN in servo design provided early guidance for a design process that combined FEM with DSL (Dynamic Simulation Language, an experimental simulation programming language) to aid in design of the actuator and servo. Engineers in the development programs for the IBM 3380 and 3390 direct-access storage subsystems began to assemble more HDA components in order to gain an improved understanding of component interaction. With stress problems, the analyst often gets results with components only; however, with dynamic problems, the results are influenced by the interaction of many parts, so a system model is often required. Several software tools contributed to this effort. A NASTRAN pre-processor and a substructuring program for combining NASTRAN superelements allowed modelers to pre- and post-process large system models. Before HDA form factors began to shrink, the hardware was complex, and the associated FE models were very large. The HDA was divided into substructures (superelements), with different groups creating different parts of the overall model. The resulting substructured HDA system model allowed design "what if's" to affect only a portion of the model, thereby saving host computer resources.

At this point, the output from the model included mode shapes and frequencies and frequency and time-domain plots for components (such as the actuator or spindle) of the system HDA model. Little effort was made to account for spindle rotational effects on the system modes. Eventually FEM and test data did provide a basic understanding of the rotational effects of the spindle assembly on the HDA system. However, most FE codes do not change the stiffness matrix to include rotational effects; they are more usually accounted for by using mode-splitting calculations.

Because of the required high clamp load, one of the most highly stressed components in an HDA is the disk clamp. Initially, linear modeling was used for the clamps, but as designs progressed, nonlinear FE modeling was required. Eventually it became necessary to include large deformations as well as geometric and material nonlinearity, along with manufacturing boundary conditions applied in sequence. The complexity of solving this type of problem requires the model to be of modest size. Large HDA system models are assumed to be linear because it would be impractical to solve large nonlinear dynamic FE models, but as drives were downsized to 3.5-in. form factors, the HDA became much simpler and the FE model smaller.

Unlike the aerospace industry, which by necessity places great emphasis on predicting stress and/or dynamics, the disk drive industry had previously supported very limited resources for FEM predictive tools, relying instead on testing.

Present FEM/design process

Except for a few stress problems relating to small components, most DASD FEM is concerned with dynamics or thermal deformation. The number of elements required to show a complex mode shape, at the high end of the frequency bandwidth, defines the mesh density. In most cases the required mesh density to capture dynamics is much lower than the mesh density required to describe a stress field. In a similar way, the mesh density required to capture thermal deformation is usually much lower than that required to describe a stress field. Because of this low mesh density, the HDA system FE model becomes practical. If traditional high mesh

[1] M. R. Hatch, internal IBM notes on using NASTRAN in servo design, General Products Division, San Jose, 1973.

densities were required, the resulting FE model would be hard-pressed to run on a workstation in today's fast-paced development environment. **Figure 1** is a CAD drawing of a rotary actuator comb; compare this drawing with **Figure 2**, which shows an FEM for the rotary actuator assembly. Since this model was used for dynamics, the mesh density is coarse. Notice that the model comprises both coarse tetrahedron (four-sided) and "brick" solid elements, along with shell elements.

There are currently fewer difficulties with FEM than in the past. However, modeling problems still exist:

- *Model simplification* It is generally not necessary to include all geometry features. Experienced modelers know which features may be eliminated from a model, thereby saving time and cost (for example, holes, notches, and fillets are often not included in a dynamics model).
- *Transfer of geometry* Often geometry from the design tool (CAD) must be transferred to the analysis tool (CAE). Programs to transfer geometry between different tool sets are not robust. Even the transfer of simple wire-frame geometry data requires time-consuming "cleanup" of drawing titles, notes, etc. before translation.
- *Test correlation* Obtaining adequate test data is a constant problem, since test data used to correlate FE models are very time-consuming to obtain. For example, the time to complete the modal analysis for an actuator can be several weeks, and the actuator is only one of several assemblies within an HDA. Modal analysis also requires special technical expertise, which is often not available. The reality is that test data are not available for all components and systems, so FE modelers make do with available data. For dynamics, the necessary test data are those concerned with mode shapes and frequencies and frequency-response functions, although time-domain data may be required. For thermal deformation, laser holography is a valuable tool for model validation because it provides full field data, not just discrete values, and it has excellent resolution. Several companies offer software tools that aid correlation between test data and FEM.
- *Excess model size* During the past year, analysis has migrated from the host to the workstation platform. Because of the limited DASD capacity available on the workstation platform, the modeler must now limit the size of the model. For FE dynamics models, the maximum practical model size depends on the number of mode shapes to be investigated. Generally, practical limits are reached with HDA models of 30000 degrees of freedom over a 1-kHz bandwidth. Much larger models tend to require more scratch-file space than is available with the workstation, and execution time can exceed one-day turnaround. In the future, as workstations gain more power, the limitations will change. At present, large nonlinear dynamics problems, such as those associated with automobile crashes, are modeled on mainframe supercomputers. For a given amount

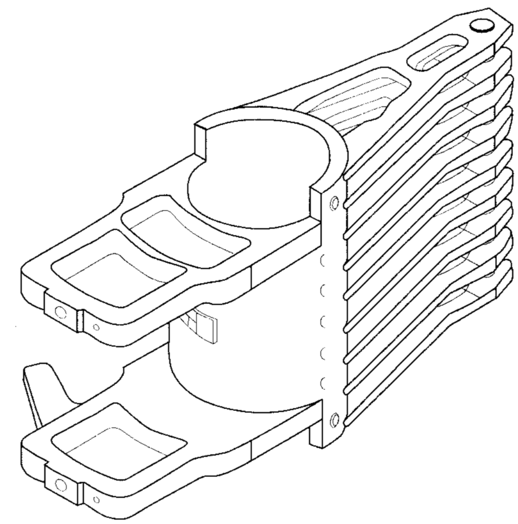


Figure 1

Rotary actuator comb.

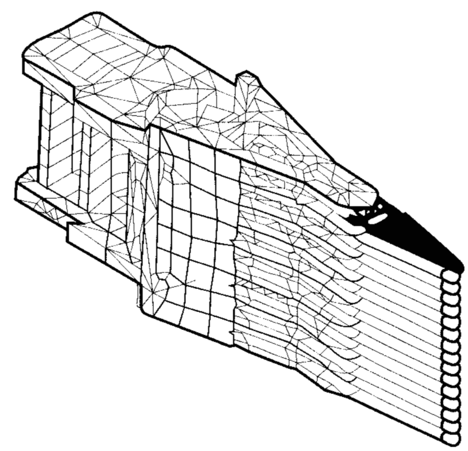


Figure 2

Rotary actuator assembly FE model.

of computing power, workstations tend to be less expensive, so we are fortunate that our HDA linear dynamics models fit the workstation profile.

- *Auto-meshers* Most codes today offer 3D automatic mesh generation of tetrahedron solid elements. These meshers often fail with complex geometry, and auto-meshing tends to produce very large models, which is inappropriate for the limited DASD resources of a workstation. Most HDA FE modeling today is done instead with powerful meshing tools, which can be controlled to create quadratic shells and six-sided "brick" solid elements, but are not fully automatic.
- *Experience with boundary conditions* The primary reason why inexperienced modelers can get misleading results is the lack of proper boundary conditions. (For example, how far should the FE model extend to account for compliance of the surrounding structure? Also, over-restraining a boundary condition often causes the model to appear much stiffer than the actual structure. Soft joints in a model, such as vibration isolators, must be carefully modeled, or key modes will be very inaccurate.) Because of the difficulty of automating the boundary conditions in an FE model, "push-button" modeling is still in the future.

Today these obstacles unfortunately limit the widespread use of FEM in many industries.

The two examples that follow demonstrate the use of FEM in the design process. The first example discusses how FEM fits into the suspension design process.

- *Three-piece suspension*

This example is related to a typical suspension development process. As usual, the design of a new component must satisfy certain requirements established by the development team. These relate to the mass and inertia, dynamic response, structure profile and merge constraints, gimbal and load beam stiffness, reworkability, cost, and a number of other factors. A brief discussion of the general guidelines for suspension design and analysis follows. Some design and analysis aspects can be assessed analytically, with good assumptions and some effort; others may be more fully understood by using FEM techniques.

Vertical, pitch, and roll stiffnesses of the suspension are usually determined by an air-bearing flying-height tolerance assessment. Dynamic excursions occur during operation because of suspension vibration and disk run-out motion, but their effect on the flying height is negligible because the air-bearing stiffness is considerably greater than that of the suspension. Suspension stiffnesses other than the above three should be as high as possible to reduce unwanted slider motion.

Operational stresses are proportional to the slider preload. Occasional stresses to be considered are related to stiction loads, jostling and shock during shipment, and crash stop accelerations. The remaining initial stresses due to metal forming must also be considered. Fatigue stresses are related to stiction and load/unload forces. Vibrational stresses during operation are usually small, but should be checked against fatigue. A detailed FE model of the entire head/suspension assembly provides both stiffness and stress data.

The suspension can buckle under stiction loads applied to the slider in the longitudinal and tangential directions. A large preload produces large stiction forces. Of the load that causes buckling or the load that makes the stress reach its maximum acceptable value, the smaller becomes the limit load in any particular direction. Thin shell elements are used extensively in the FE suspension models to provide load stress data without the added complexity of solid elements. The relative model simplicity provided by shell elements and the ability of most codes to provide nonlinear solutions allow residual stress and buckling to be predicted.

Vibration of the suspension assembly causes in-plane slider motion and flying height modulation. Torsional and sway modes are detrimental with rotary actuators. The vibration of the suspension assembly is excited by the vibration of the disk spindle, the actuator seeking motion, and particularly, the air flow inside the enclosure. The amplitude of vibration is proportional to the degree of excitation. The suspension design must compromise between high resonant frequencies, which demand a stiffer suspension, and good flying-height control, which requires low slider stiffness. The same FE model used for stiffness/loads/stress is used for dynamics. Normal-mode solutions provide mode shapes and frequencies for suspension bending and torsion modes.

Damping is necessary to reduce unwanted motions of the slider. The most common damping method for conventional suspensions is constrained-layer damping, which can be estimated using FEM. However, since this process is very time-consuming, test data are often input into the FE models.

Tolerance of shock during shipping is indicated by the onset of separation of a preloaded slider from the disk when the suspension assembly is subject to an acceleration that drives the slider away from the disk. The value of this acceleration, expressed in G's, is proportional to the preload. A FEM static global acceleration-load analysis is usually enough to compute the shock-onset acceleration of the assembly.

Sliders are usually bonded to the flexure with epoxy adhesive, which must be strong enough to take stiction loads. Because of thermal mismatch between steel and the slider material, the slider may undergo temperature-dependent bending that affects flying height. The design of the bond pad must therefore minimize slider bending. This problem can be solved analytically in a simplified form, but a more complete assessment can be made through the use of FEM.

Wiring electrically connects the slider to the computer. To reduce wire stiffness, a wire loop is provided between the slider and the load beam. The main problem is to determine the required stiffness of the loop. Nonlinear thin-rod theory is used to solve analytically for the stiffness of two wires twisted together, including the effect of the plastic insulators in contact. This stiffness is then prescribed in rod elements, and the wire loop is constructed and solved at the suspension level with FEM.

Suspensions are attached to the arms by either mount plates or weldments. The clamping force of the plastically deformed mount-plate stud is assessed with nonlinear FEM. The load beam is spot-welded to the mount plate. The compliance of these weld spots is important in assessing sway/side-bending modes of the suspension and preload stiffness. Each weld spot possesses six compliances (two bending, two axial, and two shear) that can be analytically estimated using a variation of the theory of elasticity for contact problems. Alternatively, an *ad hoc* model of the interface at the weld spot could be used. The resulting compliances are added as springs to the FE model. The model is then used to minimize the number of spot welds and optimize dynamics.

The description above summarizes the requirements and design issues for a typical suspension development process. After a particular design has been identified, product features are designed on the basis of analysis, experience, cost, and manufacturing concerns. FEM analysis, together with closed-form analysis, is repeatedly used to improve design and evaluate alternatives. Testing, particularly of dynamics and stiffnesses, corroborates calculations and helps revise the model. Linear and nonlinear statics, normal modes, and buckling analysis FEM solutions are utilized. MSC/NASTRAN and ARC Polyfem are the codes that have been employed.

- *Actuator comb assembly*

The second example is the thermal deformation of a comb/bearing-cartridge assembly. This example is the result of a team effort by testing and analysis engineers.

Diu and Lawson[2] presented methodologies and tool development for the finite element analysis of the thermal distortion in actuators caused by thermal mismatch between the comb and bearing-cartridge assemblies. Crawforth, Gong, and Young[3] correlated the analytical results of a specific actuator assembly to the experimental test results.

In their paper, Diu and Lawson demonstrated the usefulness of the finite element method in describing the mechanism responsible for the thermal mismatch between the comb and bearing-cartridge assemblies. Beginning with a simplified analytical expression to describe the thermal interaction between the comb and bearing cartridge, Diu showed the nonconservative behavior of the interface due to temperature cycling. From a parametric study, the amount of thermal distortion due to thermal cycling could be minimized by increasing the comb bore diameter and by reducing the mismatch in the coefficients of thermal expansion (CTE) for the comb and bearing cartridge, the length of the bearing cartridge, the offset angle in the bearing cartridge for the fastening screw, the angle of relief in the comb for the bearing cartridge, and the span of the arm tips from top to bottom. In the finite element phase, the interface between the comb and bearing cartridge could be represented by spring elements (leading to a linear analysis), or it could be represented by gap elements to simulate the slippage and contact forces at the interface (leading to a nonlinear analysis). The paper demonstrated that the use of spring elements cannot account for thermal slippage, and that the choice of gap elements was preferred.

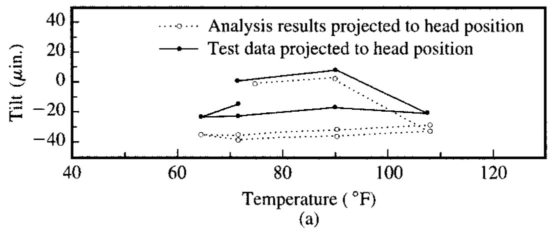
The nonlinear analysis results were demonstrated with the ANSYS finite element code [1]. The suggestion for validation of the simulation phase was pursued in the work of Crawforth et al.[3] An important finding from this work was the importance of properly selecting the values used for the coefficient of friction in the gap elements and the distribution of those values among the gap elements used to represent the comb/bearing-cartridge interface.

The simulation of the behavior of the interface between the comb and bearing cartridge assemblies depends upon many assumptions used to model the physical system. The hysteresis phenomenon at the interface can be inferred from thermal distortion test measurements obtained using holographic interferometry. The holographic interferometer is a laser-optical device that has the ability to measure, nondestructively, the relative distortion of the test parts in the microinch range. Crawforth et al. demonstrated the successful validation of the finite element model with test results for the Lightning comb and actuator assembly.

The example of the Corsair actuator is used to illustrate the use of the finite element method to represent the thermal distortion at the comb and bearing-cartridge interface. The FEM simulation was tuned by comparing

[2] Anselm C. K. Diu and Drew Lawson, "Applying FEA Methods to Study the Thermal TMR Originating in the Root of the Actuator Comb," internal IBM report, Storage Systems Division, San Jose, January 1991.

[3] L. Crawforth, K. Gong, and K. Young, "Correlation of Analytical and Experimental Study of Thermal Distortion of Lightning Actuator Assembly," internal IBM report, Storage Systems Division, San Jose, 1992.



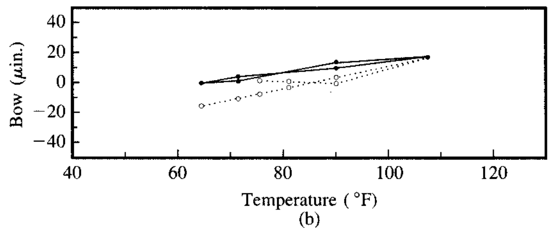


Figure 3

Comparison of analysis results with test data for Corsair comb assembly: (a) tilt vs. temperature cycling; (b) bow vs. temperature cycling.

the results to the holographic interferometry test results. The purpose of this activity was to establish a benchmark for comparison to subsequent actuator designs.

A finite element model of the Corsair comb assembly was created by using a graphics pre-processor, the CAEDS Graphics Finite Element Modeler (GFEM) [2]. GFEM was also used to create the finite element model of the sleeve of the bearing cartridge. The interface between the comb core and the bearing-cartridge sleeve was represented with gap elements. (A gap element can represent an initial gap separation or an initial contact surface which can have relative sliding.) The gap element used here is a three-dimensional interface element which can represent the normal and tangential stiffness at the interface. The pre-load of the gap elements is represented through the initial strain of a beam element used to model the attachment screw between the comb and sleeve.

The Swanson Analysis System (SAS) ANSYS [1] finite element solver was used to perform the nonlinear analysis. This code was selected for its support of a three-dimensional interface element, for its capability to represent initial strain in a fastening screw, and for its capability to define thermal slippage in the gap element as a function of temperature load cases. This last feature is particularly important because the energy losses due to the friction joints, as well as the hysteresis loop of the comb tip deformation, are dependent upon the temperature load path.

Within the ANSYS input (PREP7) file, the nodes and elements for the finite element model are listed, along with the material properties, the initial boundary conditions, and the thermal loading. In this example, the comb material was magnesium, and the bearing-cartridge sleeve was steel. The desired preload force in the set screw was defined with initial strain in the beam element used to represent the set screw. The coefficient of friction in the gap elements was adjusted for different analyses. The evaluations for both the FEM and the holographic interferometer were performed at 10° intervals, with the ambient temperature swinging from room temperature to 42°C, then down to 18°C, and returning to room temperature.

Using the holographic interferometer, displacement measurements were taken at the bearing-cartridge axis to establish a reference, and at the end of each arm tip for each temperature interval. All of the rigid-body motion due to shifting of the fixturing and the optics was analytically removed, leaving only the displacement of the arm tips relative to the bearing axis. The resulting arm-tip displacements were decomposed into TILT and BOW components by successively applying a first-order and second-order polynomial curve-fitting technique. The tables of values for both the FEM and the holographic interferometer were post-processed by the holographic data reduction code for commonality. The FEM simulation was tuned to converge on the holographic evaluation results. Refer to **Figure 3** for an analysis-to-test comparison.

This type of analysis is very important for understanding the effects on comb assembly distortion caused by relationships among the material coefficient of thermal expansion (CTE), the coefficient of friction, and the temperature. Prior to this effort, designers had little understanding of the interactions involved in this complicated phenomenon.

Future design impact of FEM on DASD

As FEM tools evolve, development time and costs will be saved. Future tools will probably include the following improvements:

- Expert system help on simplifying geometry, boundary conditions, and test correlation.
- Seamless geometry transfer between CAD and CAE.
- Robust auto-meshers with adaptable meshing features.
- Improved software and hardware that will allow nonlinear dynamics for large system models on workstation platforms.
- Extended use of P-code, which increases the order of the element to provide more accuracy.
- Integration of servo system and spindle rotational effects into the structural code.

Concluding remarks

Older DASD programs had development cycles that allowed for extensive matrix testing to solve dynamics and thermal problems. Recent programs have development cycles of 12 to 18 months and do not allow for lengthy, time-intensive testing. On one hand, software is becoming more expert, thereby speeding problem resolution. On the other hand, dynamic and thermal problems are being amplified by higher TMR, external vibration, and shock environments.

DASD development remains an involved team effort. The design engineer responsible for HDA components does not have the many skills needed for a complete understanding of all design issues. In addition to detailed design knowledge, a development program requires specialized expertise in testing and/or simulation. Future software that will allow "push-button" development coupled with minimal testing is still a long way off, but it seems evident that CAE/FEM will increase in scope and contribution to future DASD programs.

Acknowledgment

The authors wish to acknowledge and thank L. Gilliland for providing guidance and encouragement during the writing of this paper.

References

1. ANSYS®—Engineering Analysis System (Finite Element Analysis Software), Swanson Analysis Systems, Inc., P.O. Box 65, Houston, PA 15342.
2. *Graphics Finite Element Modeler*, CAEDS manual, Order No. SH52-0263-00; available through IBM branch offices.

Received March 10, 1995; accepted for publication July 26, 1996

Terry Kan *IBM Storage Systems Division, 5600 Cottle Road, San Jose, California 95193 (KAN at SJEVM5).* Mr. Kan received his B.S. and M.S. degrees in mechanical engineering from the University of California at Berkeley in 1972 and 1974, respectively. He joined IBM in 1977 and spent four years in manufacturing engineering, working in the area of process support for the 3350/3370 HDA manufacturing. This was followed by seven years in the San Jose development laboratory, working in the area of mechanical analysis and vibration testing, and three years at the Magnetic Recording Institute, also in San Jose, working in the area of advanced actuator development. Mr. Kan is currently an advisory engineer in the IBM Storage Systems Division development laboratory working in the field of dynamics and vibration analysis. He is a member of the American Society of Mechanical Engineers and a registered professional engineer in the state of California.

Drew B. Lawson *IBM Storage Systems Division, 5600 Cottle Road, San Jose, California 95193 (DREW at SJEVM5).* Mr. Lawson received his B.S.M.E. degree from the University of Michigan and his M.S.M.E. degree from the University of California at Santa Barbara. He joined IBM in 1977 and spent three years in manufacturing engineering. From 1980 to the present, he has worked in development in the area of mechanical analysis and testing. Currently Mr. Lawson is an advisory engineer working in the field of dynamics and vibration analysis using a variety of finite element codes. He is a registered professional engineer in the state of California.

Oscar J. Ruiz *IBM Storage Systems Division, 5600 Cottle Road, San Jose, California 95193 (OSCAR at SJEVM5).* Dr. Ruiz received a degree in mechanical engineering from the Catholic University, Lima, Peru, in 1982 and the M.S. and Ph.D. degrees in mechanical engineering from the University of California at Berkeley in 1989. He joined the Magnetic Recording Institute in 1989 to work on laminated arms and suspensions and other mechanical devices. Currently he is involved in the analysis and design of actuator components for disk files. Mr. Ruiz's interests include the analysis of nonlinear structures found in disk drives, and shock damage protection in disk files.

Carl F. Sermon *IBM Storage Systems Division, 5600 Cottle Road, San Jose, California 95193 (SERMON at SJEVM5).* Mr. Sermon is an advisory engineer and has completed his assignment on the use and development of laser holographic interferometry as a tool to diagnose and solve development and production design problems. He is currently a manufacturing engineer. His work in the past has included developing on-site robotic automation projects, managing a project team for a dock-to-dock material handling system, and developing a disk production surface scanning laser inspection tool. Before joining IBM eighteen years ago, Mr. Sermon worked for the General Electric Nuclear Division and Lawrence Livermore Laboratories, and was a lecturer in the Engineering Department at San Jose State University. He received his B.S.M.E. and M.S.M.E. degrees from San Jose State University.