SEPTEMBER/OCTOBER 1990

MECHANICAL ENGINEERING SYSTEMS

USING FATIGUE AND FRACTURE ANALYSIS TO OPTIMIZE DESIGN

NEW DEVELOPMENTS IN DIRECT DATA BASE CONVERSION

DYNAMIC ANALYSIS AND DESIGN SOFTWARE

AUERBACH PUBLISHERS

Integrating Finite-Element Analysis with Quasi-Static Loadings from a Large-Displacement Dynamic Analysis

PATRICK RYAN TURNER

This article discusses a process that dramatically reduces the work needed to accurately model the loading conditions of a mechanical part undergoing dynamic motion. The method involves capturing mechanical loads (both joint and inertial) from results produced by a large-displacement elastodynamic analysis program (e.g., Mechanical Dynamic Inc's ADAMS) and directly applying them to a finite-element model. This procedure allows the analyst to perform a quasi-static equilibrium analysis without regard to actual boundary conditions. Errors are minimized and productivity is increased by automating the transfer of complete and consistent data between different analysis types. In fact, total design analysis accuracy is increased over standard practices because approximation in the transfer of load information is eliminated.

The article also outlines the theory required to understand the process, along with suggestions for checking the accuracy and consistency between the ADAMS model and the finite-element model.

Structural engineers are plagued with the problem of where to get loads for finite-element analysis. Once loads are available, the question remains as to how accurately they represent the actual operating environment of a given part. This accuracy depends greatly on the origins of the loads. Loads typically are obtained through one of the following methods:

- Strain gauging, which sometimes requires altering the structure.
- Accelerometer measurement, which affects mass.
- Photoelastic stress analysis (the static test only).
- Kinematic or dynamic simulation, based on approximate input.
- Historical value measurement (derived from the previously mentioned tactics) for parts used in similar situations.

These methods usually result in some approximation in the loading of a given part. Even if the engineer is willing to accept this level of approximation, the actual application of the loads to the finite-element model is a tedious and error-prone task. Manual data entry of loads and coordinate transformations are only two examples of error origins. In

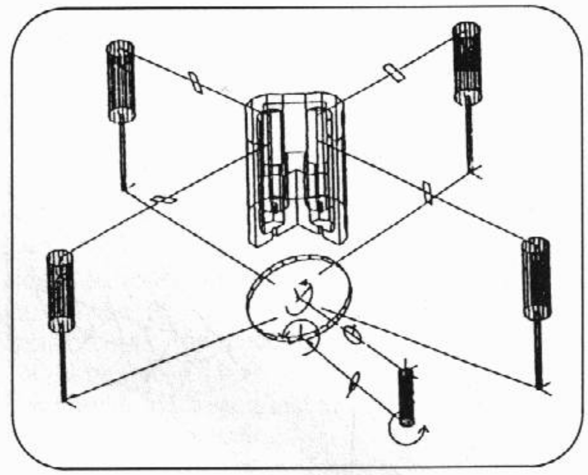
***Patrick Ryan Turner** is a manager in the product development department at Mechanical Dynamics in Ann Arbor MI. He holds an MS in mechanical engineering from Massachusetts Institute of Technology and a BS in mechanical engineering from Michigan Technological University. He has written numerous papers that have been presented at industry conferences.*

addition, although parts may be in dynamic motion, inertial loads are typically ignored, and only joint forces or externally applied point loads are included in the finite-element analysis.

However, inertial loading on a dynamically moving part may significantly affect the part's structural integrity and must be considered. This can be done through the application of the quasi-static principle, which implies that any part moving through space is in quasi-static equilibrium at any given instant in time if all loading, both external and inertial, is considered.

This article focuses on the transfer of loads from a large-displacement analysis program to a finite-element model. Two graphic modeling systems are used. First, the Schlumberger Inc MECHANISMS program is a graphic interface to the ADAMS analysis program. Second, the Schlumberger GRAFEM (Graphic Finite Element Modeling) program interfaces to the MacNeal-Schwendler Corp MSC/NASTRAN and the IFAD (Integrated Finite Element Analysis for Design) finite-element analysis program by Schlumberger. IFAD is internal to GRAFEM. Other systems can be used, however, because the method is general and not restricted to any particular finite element or finite-element analysis code.

Exhibit 1. ***The MECHANISMS Model of the Stirling Engine***



BUILDING THE ADAMS AND FINITE-ELEMENT MODELS

Quasi-static loading of a finite-element model using results from a kinematic or dynamic simulation provides the engineer with a consistent and accurate method to structurally analyze any component of a dynamic system. For this article, a dynamic system was needed that contained both external interface loads and inertial loads caused by motion. The system selected was the Stirling Engine described in the ADAMS applications manual. The Stirling Engine, shown in Exhibit 1, is made up of four pistons, the swash plate (which makes contact with the base of the piston rods), the base center shaft (as the pistons fire in succession, the shaft is forced to rotate), and the engine block (in which the pistons reciprocate). Some will recognize this mechanism as a variable-displacement pump.

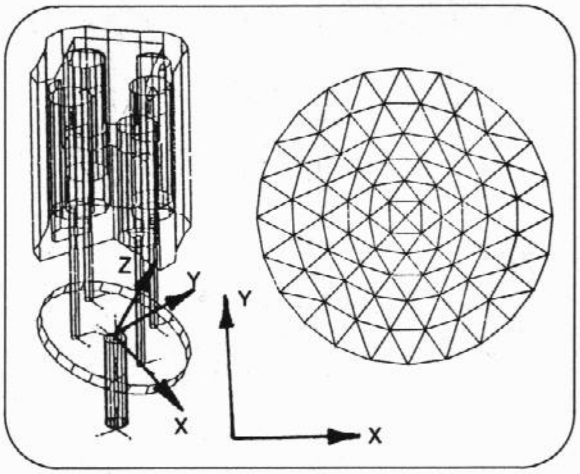
The ADAMS model of the Stirling Engine is built using the MECHANISMS graphic interface to ADAMS. The model includes parts, joints, forces, and generators. Joints and generators are indicated in Exhibit 1 by the graphics symbols connecting the exploded-part geometry. While the MECHANISMS parts are created, mass properties of the various components must be input to the system. The accuracy of the mass properties will determine the success of transferring the quasi-static loads. For this reason, a solids modeler is used to create the geometry for each of the Stirling Engine components. The same solids modeler is used to precisely calculate the mass properties of each mechanism component. Once calculated, these mass properties are in the data base and are automatically applied to the appropriate part as the MECHANISMS model is built.

Next, the finite-element model of the swash plate is constructed. The GRAFEM program is used to create the model using two-dimensional plate elements. The solid modeler geometry is used to define the boundaries of the finite-element mesh.

The Discrepancy Between Coordinate Systems

The finite-element analyst and the mechanism dynamicist are usually not the same person in an engineering organization. Therefore, the finite-element model may be built using a completely different coordinate system from that used to generate the mechanism part (see Exhibit 2). To accommodate this situation, the difference between the coordinate systems can be identified in the modeling systems (i.e., MECHANISMS and GRAFEM) and the appropriate transformation applied by the system to the loads during the load transfer process. This allows the geometry of the mechanism part and the finite-element mesh to be created in coordinate systems that

Exhibit 2. *The Finite-Element Model of the Stirling Engine and Swash Plate Using Different Coordinate Systems*



are accessible to the respective analyst. The next section discusses why it is important that similar geometry be used for both the mechanism part and the finite-element mesh.

Mass Property Consistency

As a part moves through space, it is subject to loads that depend on its mass and mass distribution. These loads, often called inertial loads, are caused by acceleration, spin velocity, and gravity fields. To understand how mass properties affect quasi-static load transfer, it is useful to examine how they are represented in the large-displacement and finite-element analyses.

For rigid-body analysis in ADAMS, mass properties are represented as single values about the center of mass or center of inertia. Single values, like polar moments of inertia, products of inertia, and mass, are input by the user for each part of the mechanism. A part's ability to accelerate or move through space is affected by mass properties. Thus, the resultant joint forces, accelerations, velocities, and displacements for a given part are driven by mass properties. Therefore, the accuracy of the results for a rigid-body, large-displacement simulation depends on the mass property distribution. These results (accurate or inaccurate) are input to the finite-element model.

Distributed Mass Properties

In finite-element analysis, each element of a part contributes to the mass matrix. The size and location of each finite element in the model dictate the mass properties of a modeled part, which are calculated internally or implicitly during the finite-element analysis. Only a few external values (e.g., element thickness, some beam element properties, and density) are entered to provide mass property information. As a result, how closely the elements conform to the actual part geometry dictates whether inertial loading will accurately affect the part.

Mass Property Impact

As discussed in the previous section, analysis results may become distorted because there is typically no common origin for mass properties derived through finite-element and large-displacement analysis. For example, if the lumped-mass properties input to ADAMS are in error, load and motion results will not be accurate. If load and motion results are not accurate, the load input to finite-element analysis will not accurately represent the part environment.

In addition, the finite-element analysis calculates distributed-mass properties essentially on the basis of conformance to part geometry. If the geometry is not adequately traced by the finite elements, the inertial loads will not have the proper structural effect. Such mass property inconsistencies can render the finite-element and ADAMS models incompatible and analysis results inaccurate.

Ensuring Consistency

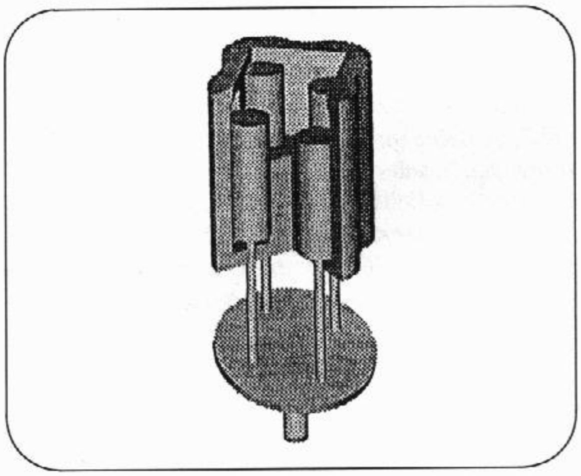
A rigorous way to ensure mass property consistency is to analyze a preliminary finite-element model. The user then determines whether the mass properties calculated by the finite-element analysis are the same as those input to the ADAMS model. This process is usually not practical or time efficient.

A more efficient method is to use a solids modeler to generate geometry for both the MECHANISMS model (i.e., ADAMS) and the finite-element modeler (i.e., GRAFEM). The solids modeler can calculate a full and precise set of mass properties for use in the ADAMS analysis. Then, the solids model geometry is used to create the finite-element model, ensuring consistency. The SOLIDS MODELER program by Schlumberger was used in this manner for the Stirling Engine example (see Exhibit 3).

In summary, mass properties are obtained by the dynamicist through various sources, including:

- Historical values from previous similar designs.
- Mass analysis departments.
- Manual calculations (the most tedious and time-consuming process).
- A preliminary finite-element analysis.

Exhibit 3. ***The Solid Model of the Stirling Engine***



All but the last of these sources generally provide only the axial inertias, leaving the products of inertia to be estimated. In addition, these techniques are time-consuming and error prone and usually lead to approximation, which provides less accurate results to the engineer. More important, when the input for the finite-element analysis depends on the output of the large-displacement analysis, these errors of approximation or inconsistency build up and proliferate through subsequent analyses. This can be avoided by using a solids modeler as a focal point for consistency in the model. Using a solids modeler provides an accurate and complete set of mass properties for the large-displacement analysis and geometry for the finite-element analysis.

The Load-Transfer Procedure

Once the mechanism model is complete, an ADAMS input file is generated by the MECHANISMS program. The large-displacement analysis is then performed by ADAMS and results are retrieved into the MECHANISMS data base. In the case of the Stirling Engine, motion has been simulated and the structural impact on the swash plate is analyzed when piston number three is at maximum acceleration (see Exhibit 4). The engineer selects this output-time step by making it the current display. The user determines whether the same coordinate system (i.e., ADAMS local-part reference frame) has been used in the generation of the finite-element model. If not, the modeling system performs a coordinate transformation on the loads during the transfer process.

At this point, the engineer should ensure that mass properties are consistent and that the units used in the ADAMS analysis are the same as those to be used in the finite-element analysis. In the Stirling Engine example, the units may be changed at any time during the modeling or result review process. This change will be reflected in any load data generated by the system.

The motion data from the dynamic analysis is handled according to the type of finite-element-analysis code employed. MSC/NASTRAN supports only global model accelerations; IFAD supports only per-element accelerations. The components of motion are transformed into an acceptable general format for each case. The motion types are:

- Body forces due to gravity.
- Inertial loadings affected by translational and angular acceleration.
- Forces due to angular (spin) velocity.

When MSC/NASTRAN is employed as the finite-element-analysis code, motion is translated into RFORCE and GRAV cards. The body forces due to gravity and translational accelerations are written out as GRAV cards; the angular accelerations and spin velocities are written out as RFORCE cards. These motions are applied to the finite-element model in separate SUBCASEs. Then, a combination SUBCASE is created with a LOAD card to generate the final results and put the part in quasi-static equilibrium.

When loads are transferred to IFAD, the distance from the center of each finite element to the finite element-model coordinate system origin must be

Exhibit 4. ***The Simulated Motion of the Stirling Engine at the Desired Time Step***

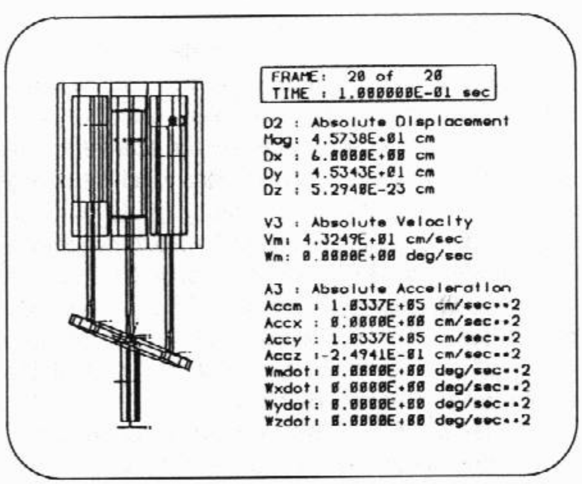
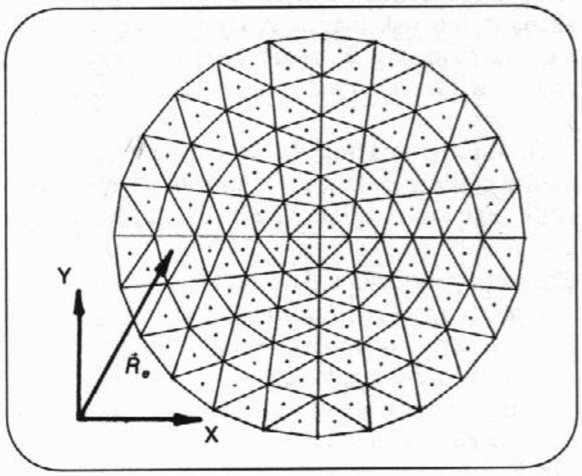


Exhibit 5. ***Position Vectors for the Finite-Element Model During Load Transfer to IFAD***



found (see Exhibit 5). This is necessary because the motion of the mechanism is specified as single values only about the local part reference frame. The inertia loading must be distributed to each finite element of the part using R_e. All motion types received from the ADAMS analysis by the MECHANISMS program are combined and applied as a single linear acceleration to each element. This requires the system to use the centroidal location vector of each element and take the appropriate cross products for each motion type. This process requires thousands of vector calculations to distribute the motion loads to a moderately sized finite-element model. Automating this process is the only practical way to apply quasi-static loading.

Advantages

A subtle but important advantage of this load transfer process is that the load application to the finite-element model is independent of the element geometry used. As long as mass property consistency is maintained, the engineer may use beam, shell, or solid elements to successfully perform the quasi-static finite-element analysis. This provides much flexibility in the generation of finite-element models. In IFAD, only linear accelerations are applied to the finite-element model. For MSC/NASTRAN, only RFORCE and GRAV cards need to be created. The inertial load effects (including the density and volume of each element) are calculated as an internal part of the finite-element-analysis process.

Another nuance of this process is that the complete finite-element model can be formatted to any finite-element-analysis program supported by the finite-element modeler. This is because the loads transferred are written to the finite-element modeling program rather than written to any program-specific format.

Finally, the quasi-static load transfer from the MECHANISMS program supports two-dimensional problems. Results produced by DRAM (Dynamic Response to Articulated Machinery) can be used to load a fully three-dimensional finite-element model. The loading will be completely in-plane with the mechanism and produce no ambiguities in the process.

Transfer of Loads

All external loads and joint forces are placed at nodes in the finite-element model. These external forces can be caused by any force element (e.g., spring, damper, beam) that is present in the ADAMS analysis. During this part of the load transfer process, nodes are created at the proper location in the finite-element model (see Exhibit 6). Concentrated loads and moments are then placed on those nodes. The user is responsible for connecting all the unattached loaded nodes to the rest of the finite-element model. Because several methods of connection can be used and the information that the user expects from the finite-element analysis varies (parabolic or cyclic distribution of these concentrated loads along a surface may be used), the transfer process cannot assume how the nodes should be connected to the model, leaving the decision to the user.

Exhibit 6. ***The Finite-Element Model of the Swash Plate with Applied Nodes***

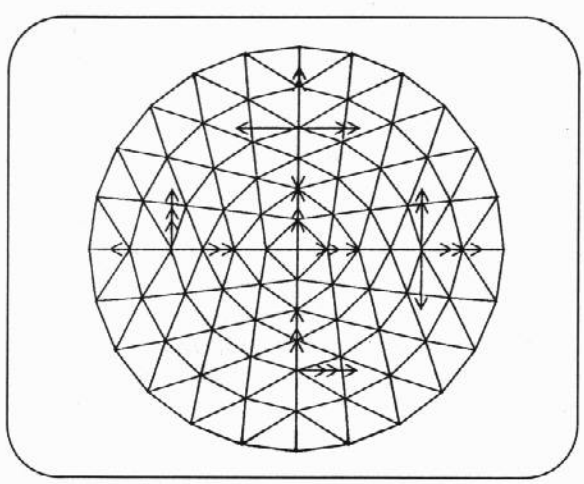
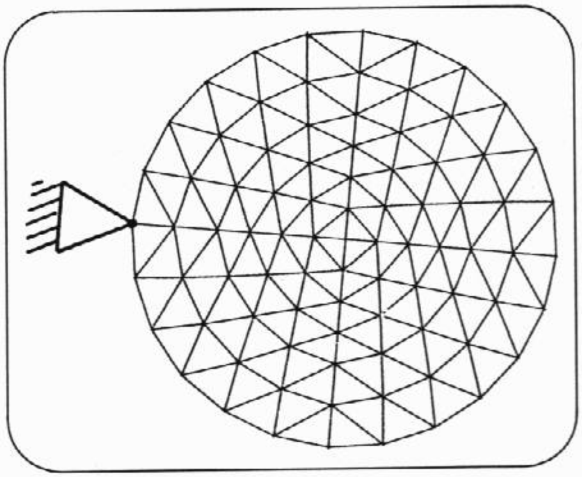


Exhibit 7. ***The Finite-Element Model of the Swash Plate with a Single-Point Restraint***



Preventing Rigid-Body Motion

After all the loading has been applied, a rigid-body restraint must be placed on the finite-element model, as shown in Exhibit 7. Without a rigid-body restraint, it is impossible to put the finite-element model precisely in equilibrium because of round-off error in the computer. For plate models, at least one node must be restrained in all six degrees of freedom. This one single-point restraint will prevent large rigid-body motions or, in more formal terms, will keep the finite-element stiffness matrix from being singular. This restraint should be placed on a node in a noncritical area of the model to avoid distorting the results in a critical area.

Care must be taken to apply only those restraints necessary to prohibit rigid-body motion. If multiple single-point restraints are needed (i.e., when solid elements are used), they should be located directly adjacent to each other. If not, deformation caused by loading may be prohibited, distorting the results.

For example, a slender rod is rotated about one end and has a substantial weight at the other end. Assuming that solid elements are used in the finite-element analysis, two or three restraints are required to prevent rigid-body rotations. The user, in error, locates one restraint on one end of the rod and two on the other end, preventing the bar from elongating. In this way, the application of the restraints inhibits deformation of the rod and thus prevents a correct solution.

Checking Quasi-Static Conditions

The finite-element analysis can be run using any finite-element-analysis program (e.g., MSC/NASTRAN and IFAD) supported by the finite-element modeling system. Both concentrated and acceleration loads must be supported by the analysis program.

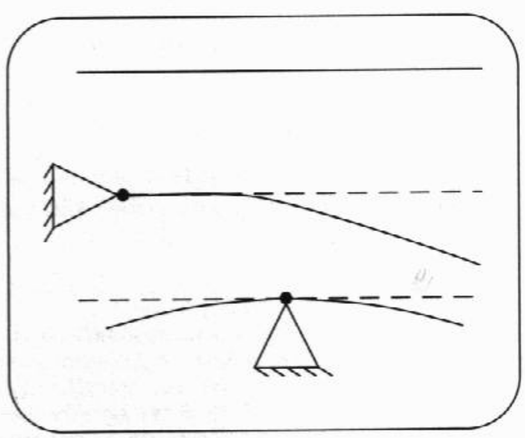
After the analysis, the reaction forces at the restraints should be reviewed. If they are close to zero as a percentage of the total loading on the model, the part is in quasi-static equilibrium. In other words, all forces are essentially internal to the part.

If the reaction forces are great, the mass properties used in the large-displacement analysis may not be consistent with those internally calculated by the finite-element analysis. This means the restraints have generated reaction forces that compensate for the imbalance. Another possibility is that the coordinate system used in the load transfer process may not be consistent with the large-displacement analysis and the finite-element model.

Considerations for Finite-Element Postprocessing

The displayed shapes of the deformed finite-element model will differ depending on which node is restrained. The restraints serve as the inertial or global reference from which all other displacements are calculated in the finite-element analysis. Therefore, animated shapes of deflection appear significantly different, depending on where this reference is located on the finite-element model (see Exhibit 8).

Exhibit 8. ***The Edge View of the Swash Plate Depicts Two Restraint Locations for the Same Loading Condition***



CONCLUSION

The process outlined in this article shortens the time required to perform analysis. It also reduces errors in design geometry and analysis input, allowing greater consistency across analysis disciplines. The step-by-step process provides a means to store design history, which is proving to be very important in today's mechanical engineering industry. Most important, automation of the tedious and error-prone tasks provides a structured design environment for inexperienced engineers and allows the engineer to focus on the design and not the tedium of the task at hand. ❑

Acknowledgments

The author would like to acknowledge the efforts of Michael Collingridge, Mitchell Walsh, and Bradley Wilkin of Schlumberger Technologies, CAD/CAM Division. Their efforts contributed to the development of this article.