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

by

Patrick Ryan Turner
Engineering Manager of Mechanical Computer Aided Engineering

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

Presented at:

The MacNeal-Schwendler Corporation

MSC World Users Conference

"CAE Tool for the 1990's and Beyond"

March 13-17, 1989

Abstract

A method for capturing mechanical loads (both joint and inertial) from results produced by a large-displacement elasto-dynamic analysis program (e.g. ADAMS) and directly applying them to a finite element (FE) model is presented. This procedure allows the analyst to perform a quasi-static equilibrium analysis without regard to actual boundary conditions. Errors are minimized and productivity 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 importance of model consistency (i.e. mass properties and coordinate systems) required between the two analysis types, and how to assure it, is presented. The method for transferring the mechanical and inertial loads was found to be independent of the particular element type used in the finite element analysis. The theory required to understand the process will be briefly outlined along with suggestions for checking the accuracy/consistency between the ADAMS model and the FE model. The focus of this paper is a process that dramatically reduces the work needed to "accurately" model the loading conditions of a mechanical part undergoing dynamic motion.

1 INTRODUCTION

In the past Structural Engineers were plagued with the problem of where to get loads for finite element (FE) analysis. Once those loads were available the question still remained as to how accurately they would represent the actual operating environment of a given part. This accuracy depends greatly upon the origins of the loads. Loads have been typically obtained through one of the following methods:

- strain gauging (sometimes required changing the structure and hard to locate in desired positions),
- accelerometer measurements (affects mass and hard to locate in desired positions),
- photo-elastic stress analysis (static test only),
- kinematic or dynamic simulation (based on approximate input), or
- historical values (derived from the previously mentioned tactics) for parts used in similar situations.

This usually resulted in some level of approximation in the loading of a given part. Even if the engineer is willing to accept this level of approximation, actually applying the loads to the finite element model is a tedious and error prone task. Manual data entry (typing) of loads and coordinate transformations are only a couple examples of error origins. Also, even though these parts may be in dynamic motion, inertial loads are typically ignored, and only joint forces or externally applied point loads are considered in the finite element analysis.

Actually, inertial loading on a dynamically moving part may have a significant affect on the structural integrity of the part. Therefore, for complete loading on a dynamically moving part, inertial loads must be considered. This can be done through the application of quasi-static principles. The quasi-static principle implies:

> *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 paper will focus on the transfer of loads from a large displacement analysis program to a finite element model. The method employed involves two graphic modeling systems. First, the MECHANISMS^{TM} program is a graphic interface to the ADAMS^{TM} analysis program. Second, the GRAFEM^{TM} (GRAphic Finite Element Modeling) program interfaces to MSC/NASTRAN^{TM} and the IFAD^{TM} (Integrated Finite element Analysis for Design) finite element analysis program. IFAD is internal to GRAFEM. However, the method is general and not restricted to any particular finite element type, or any specific finite element analysis code. Consistency of the mathematical models represented in the large displacement simulation and in the finite element model is important so that quasi-static equilibrium can be achieved for the desired part. A procedure is also outlined to verify that the analysis and load transfer have been successful and the part is actually modeled in quasi-static equilibrium.

2 DISCUSSION

Quasi-static loading of a finite element model via 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 the purpose of this paper it is desirable to select a dynamic system that contains both external interface loads and inertial loads due to motion. The dynamic system selected to demonstrate

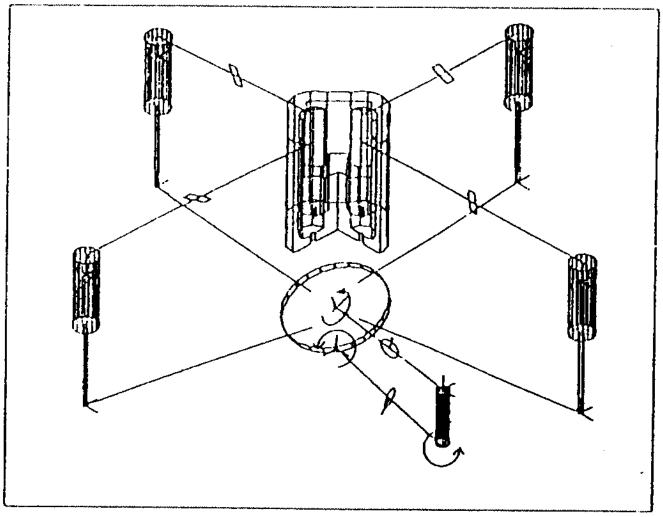

Figure 1: The MECHANISMS model of the Stirling Engine shown in exploded form

the quasi-static loading principles is the Stirling Engine given in the ADAMS Applications Manual. The Stirling Engine shown in Figure 1 is made up of various components. It has four (4) pistons, a swash plate which the base of the piston rods are in contact with, 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[1].

2.1 Building the ADAMS and Finite Element Models

The ADAMS model of the ‘he 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 Figure 1 by the graphic symbols connecting the exploded part geometry. While creating the MECHANISMS parts, mass properties of the various components must be input to the system. The accuracy of the mass properties will have significant impact on the success of transferring the quasi-static loads. For this reason, a solids modeler was used to create the geometry for each of the Stirling Engine components. The same solids modeler was used to precisely calculate the mass properties of each mechanism component. Once calculated, these mass properties are in the database and are automatically applied to the appropriate part while the MECHANISMS model is built. The completeness and accuracy of these mass properties and their impact on the FE analysis will be discussed in section 2.3.

Next, the finite element model of the swash plate is constructed. In this case, the GRAFEM program is used to construct a finite element model using 2D plate-type elements[2]. The solid modeler geometry is used to define the boundaries of the FE mesh. The transfer of quasi-static loads is independent of the geometric type of elements used in the finite element model as discussed in section 2.4.1.

2.2 Discrepancy Between Model Coordinate Systems

The finite element analyst and the mechanism dynamicist are usually not the same person within 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 as shown in Figure 2. To accommodate this situation, the user can identify the difference between the coordinate systems in the modeling systems (MECHANISMS and GRAFEM) and the appropriate transformation is 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 are convenient for the respective analyst. In general, even though it may be convenient to use these different coordinate systems, the next section

[1]The angle of the swash plate can be changed so the pistons will have a different displacement.

[2]This FE model could be constructed just as well with solid (or brick) type elements.

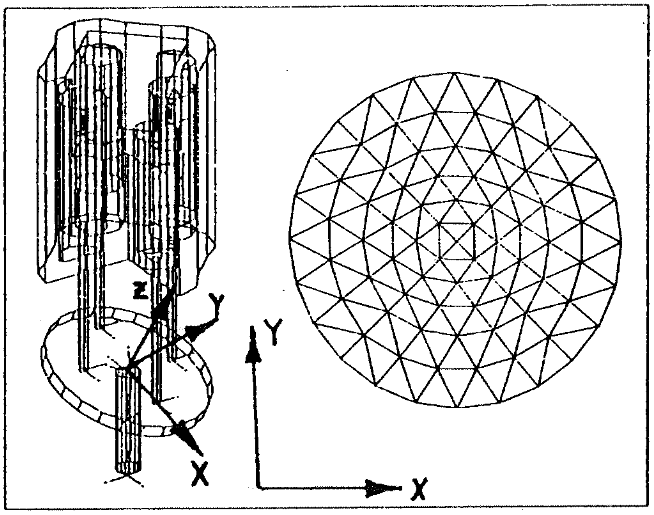

Figure 2: The ADAMS part and the FE model of the swash plate may have different coordinate systems

will discuss why it is important that similar geometry be used for both the mechanism part and the finite element mesh.

2.3 The Importance of Mass Property Consistency

As a part is moving through space, it is subject to loads that are dependent on its mass and mass distribution. These loads are often referred to as "inertial loads" and 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 these properties are represented in the large displacement analysis and the finite element analysis.

2.3.1 Lumped Mass Properties in ADAMS

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 particular mechanism part's ability to accelerate or move through space is affected by mass properties. Thus, the resultant joint forces, accelerations, velocities, and displacements (i.e. simulation results) for a given mechanism part are very much dependent on 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 used as input to the FE model.

2.3.2 Distributed Mass Properties in Finite Element Analysis

In finite element analysis each individual element which makes up the entire part contributes to the mass matrix. As a result, the mass properties of a modeled part are dependent upon the size and location of each individual finite element within the model. The overall part mass properties are calculated internally or implicitly during the finite element analysis process. There are only a few external values (like element thickness, some beam element properties, and density) that the user will enter to provide mass property information. Therefore, how closely the elements conform to the actual part geometry will dictate if inertial loading will accurately affect the part.

As seen above, the origin of mass properties and how they are represented for FE analysis and Large Displacement (LD) analysis are quite different.

2.3.3 Mass Property Impact on Analysis

Analysis results may become distorted because there is typically not a common origin for FE and LD analysis mass properties as discussed above. Case in point, if the lumped mass properties the user inputs to ADAMS are in error, load and motion results will not be accurate. If load and motion results are not accurate, load input to FE analysis will not represent the true part environment.

In addition, the FE analysis calculates distributed mass properties essentially based on conformance to part geometry. If geometry is not adequately traced by the finite elements, the inertial loads will not have the proper structural effect.

These types of mass property inconsistencies can cause the FE and ADAMS model to be incompatible and analysis results will be inaccurate. Therefore, a procedure to assure that mass property consistency is provided throughout the modeling process is discussed next.

2.3.4 Assuring Consistency

A somewhat rigorous way to assure mass property consistency is to analyze a preliminary finite element model. The user then checks if the mass properties calculated by the FE analysis are the same as those input to the ADAMS model. This is usually not practical or time efficient.

A method more consistent with the design process is to use a solids modeler to generate geometry for both the MECHANISMS (i.e. ADAMS) model and the finite element modeler (GRAFEM). The solids modeler can calculate a full and precise set of mass properties that can be used in the ADAMS analysis. Then the solids model geometry (some time later perhaps) is used to create the finite element model. Therefore, mass property consistency is assured [3]. The SOLIDS MODELER^{TM} program was used in this manner for the Stirling Engine example (refer to Figure 3).

[3]FE mesh density and geometric element type can have an impact on mass property consistency; some caution is needed.

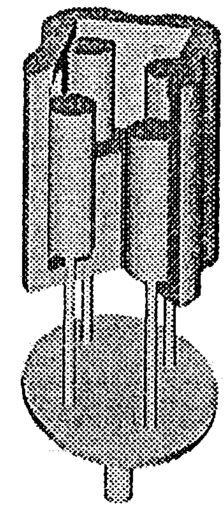

Figure 3: Solid model of the Stirling Engine using the SOLIDS MODELER^{TM} program.

2.3.5 Mass Property Consistency Summary

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

- historical values from previous similar designs,
- mass analysis departments (which never seem to calculate mass for an individual part but only on a sub-assembly basis),
- hand calculations (the most tedious and time consuming process), and
- finally as mentioned earlier, the use of a preliminary finite element analysis.

All but the last of these tactics will usually give an engineer only the "axial inertias" leaving the products-of-inertia to be estimated. These techniques are time consuming and/or error prone, and usually lead to approximation which provides less accurate results to the engineer. More importantly, 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. To avoid this proliferation of error, the solids modeler can be used as a focal point for consistency in the model. Use of the solids modeler will provide an accurate and complete set of mass properties for the large displacement analysis and geometry for the finite element analysis.

A way to check for mass property consistency will be discussed in section 2.6 through a simple check of the finite element results. Next, the load transfer process will be described in a step-by-step sequence.

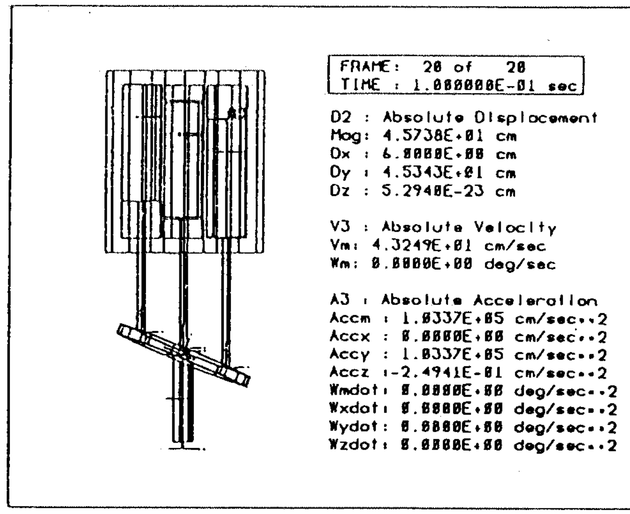

Figure 4: The motion of the system at the desired time step for the quasi-static load transfer.

2.4 The Quasi-Static Load Transfer Procedure

Once the mechanism model is complete, an ADAMS input file is generated by the MECHANISMS program. Then, the large displacement analysis is performed by ADAMS and results are retrieved into the MECHANISMS database. In this case, the motion of the Stirling Engine has been simulated and the structural impact on the swash plate is analyzed when piston number 3 is at maximum acceleration (see Figure 4). The engineer selects this output time step by making it the current display. Next, the user will identify if 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 will do a coordinate transformation on the loads during the transfer process.

At this point the engineer should assure that mass properties are consistent (refer to section 2.3), and that the units used in the ADAMS analysis are the same as those to be used in the finite element analysis. In the case of the Stirling Engine modeled in the MECHANISMS program, 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.

Next, depending on what type of capabilities the FE analysis code has, the motion data from the dynamic analysis is handled differently. MSC/NASTRAN supports global model accelerations only, and IFAD supports per element accelerations only. 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 due to translational and angular acceleration, and
- forces due to angular (spin) velocity.

When MSC/NASTRAN is employed as the FE analysis code the motion of the part is translated into RFORCE and GRAV cards. The body forces due to gravity and translational accelerations are transformed and written out as GRAV cards. The angular accelerations and spin velocities are written out as RFORCE cards. These motions are applied to the FE model in separate SUBCASEs. Then, a combination SUBCASE is created with a LOAD

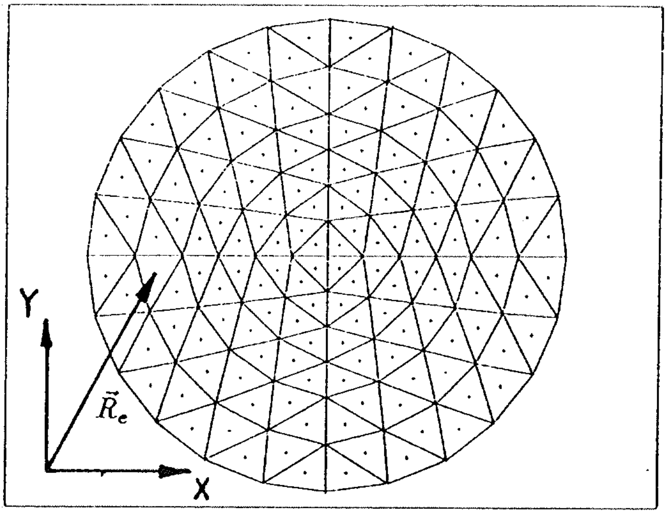

Figure 5: The position vectors for the elements in the FE model are needed during the load transfer process to programs like IFAD.

card to generate the final results and put the part in quasi-static equilibrium.

In the case of transferring loads to IFAD, the distance from the center of each finite element to the finite element model coordinate system origin must be found ($\vec{R}_e$, see Figure 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 $\vec{R}_e$. 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, $\vec{R}_e$, and take the appropriate cross products for each motion type. Here, thousands of vector calculations are required to distribute the motion loads to a moderately sized finite element model. This demonstrates that automating this process is the only practical way to apply quasi-static loading.

2.4.1 Advantages to Current Implementation

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 type used. As long as mass property consistency is maintained, the engineer may use beam type, shell type, or solid type elements to successfully perform the quasi-static FE 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 card need to be created. The inertial *load* affects (taking into account the density and volume of each element) are calculated internal to the finite element analysis process.

Another nuance of this process is that the complete FE model (loading condition and the finite element model) can be formatted out to any FE analysis program supported by the FE modeler. This is because the loads transferred are written to the FE *modeling* program rather than being written out to any FE-*analysis*-program-specific format.

Finally, the quasi-static load transfer from the MECHANISMS program also supports 2D problems. Results produced by DRAM[4] 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.

2.4.2 Transfer of Joint and Other Point Type Loads

Next, all external loads and joint forces are placed at nodes in the finite element model. These external forces can be caused by any force-type element (e.g. spring, damper, beam,

[4] DRAM^{TM} (Dynamic Response to Articulated Machinery) is a product of Mechanical Dynamics Inc. and supports the simulation of planar mechanisms.

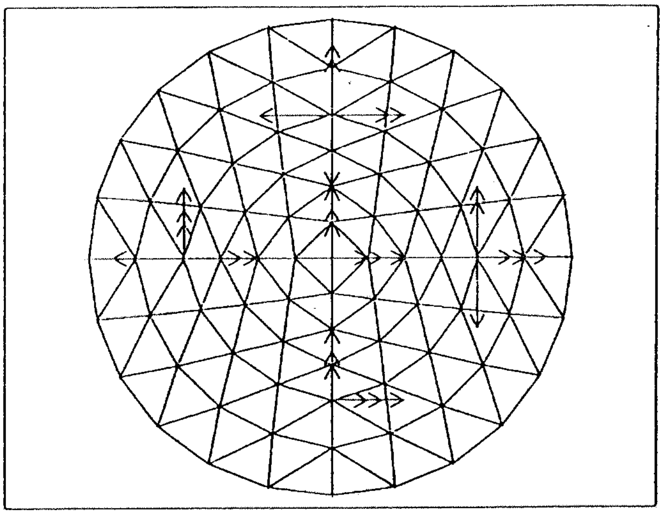

Figure 6: The concentrated forces and moments are placed on newly created nodes and the user merges them with the FE model.

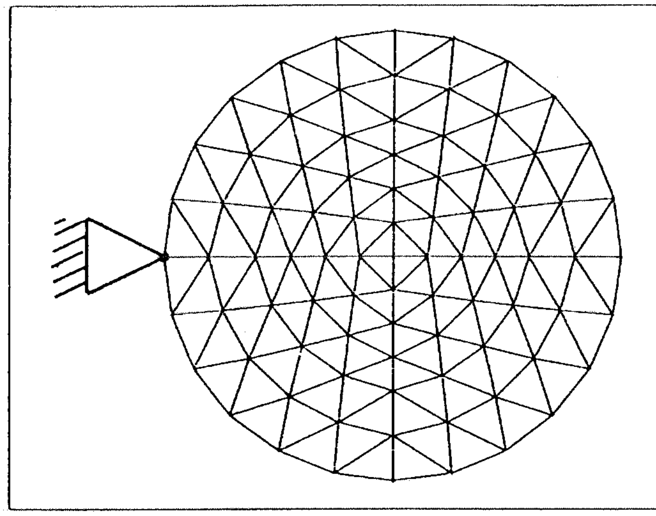

Figure 7: The finite element model is restrained to prevent rigid body motion.

etc.) that is present in the ADAMS analysis. During this part of the load transfer process nodes are "created" at the proper location in the FE model (i.e. joint centers, etc. See Figure 6). Then, concentrated loads and moments are placed on those nodes. The user is responsible for connecting all the unattached loaded nodes to the rest of the finite element model. This is left to the user because several methods can be used to connect these nodes to the model depending upon what kind of questions prompted the FE analysis. These methods include using: rigid springs, beam elements, plate elements, etc. Depending upon the information that the user expects from the finite element analysis, some form of parabolic or cyclic distribution of these concentrated loads along a surface may be used. For these reasons there is no general assumption that can be made as to how these nodes should be connected to the model. Therefore, the process is left to the user.

2.5 Preventing Rigid Body Motion

Now all the loading has been applied to the finite element model. One step remains before the FE analysis can proceed. A rigid-body restraint must be placed on the finite element model (see Figure 7). This is necessary because it is impossible to put the finite element model precisely in equilibrium due to round off error in the computer. Therefore, for plate models at least one[5] node must be restrained in all six degrees of freedom. This one single point restraint will keep the part from large rigid-body motions, or in more formal terms will keep the FE stiffness matrix from being singular. This restraint should be placed on a node in a non-critical area of the FE model to avoid distorting the results in a critical area.

Care must be taken to apply only the *necessary* restraints to prohibit rigid-body motion. If multiple single point restraints are needed (i.e. solid elements are used), they should be directly adjacent to each other. If not, some deformation due to the loading may be prohibited which would distort the results.

For example, a slender rod is rotated about

[5]For models made with solid elements 2 or 3 nodes must be restrained to prevent rigid-body rotations because solid elements do not have rotational degrees of freedom. Also, some MSC/NASTRAN plate elements do not have stiffness in the θ_z direction. Therefore, more than one node may need to be restrained.

one end and has a substantial weight at the other end. Assume solid elements are being used in the FE 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. Thus, the bar is prevented from elongating. In this way the application of the restraints has prevented getting a correct solution.

2.6 Checking that Quasi-Static Conditions Have Been Met

Next, the finite element analysis can be run using any finite element analysis program that is supported by the FE modeling system. Both concentrated and acceleration type loads must be supported by the FE analysis program, such as MSC/NASTRAN and IFAD.

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

If this reaction force is large, several things could be the cause. 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. Also, the coordinate system used in the load transfer process may not be consistent between the large displacement analysis and the FEM model.

2.7 Considerations for FE Post Processing

The displayed shapes of the deformed finite element model will look different depending upon which node is selected to be restrained on the finite element model. This restraint(s) serves as the inertial or global reference from which all other displacements will be calculated in the FE analysis. Therefore, animated mode shapes of deflection look significantly different depending where this reference is on the FE model (see Figure 8).

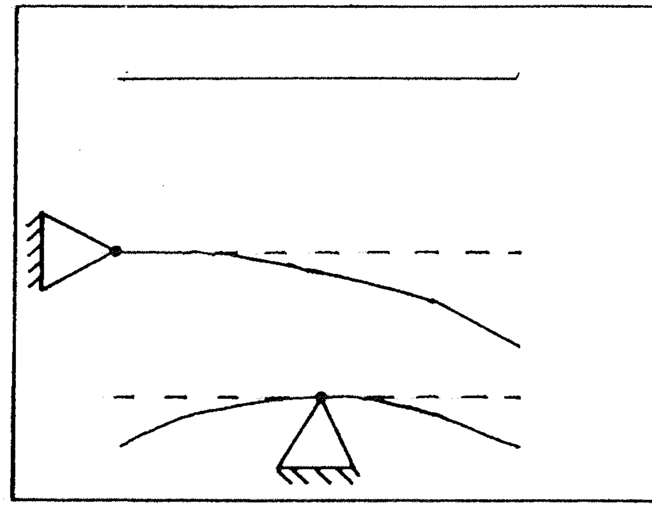

Figure 8: The deformed finite element model output is dependent on which node(s) are restrained. The edge view of the swash plate in bending is shown depicting two different restraint locations for the same loading condition.

3 CONCLUSION

Generation of a quasi-static loading for a finite element analysis using results from a large displacement analysis via a direct connection between the modeling packages has been shown. This would be impractical to do by hand because in some cases thousands of vector calculations are required to do the quasi-static load transfer. In addition, this load transfer can be done at any or all output time steps of the large displacement analysis.

It has been shown that model consistency is very important for this load transfer process to be successful. Particularly, mass property consistency must be assured between the large displacement model and the FE model. This consistency can be assured if a solids modeler is used as the focal point of the modeling process.

The load transfer technique is applicable regardless of finite element geometry type used. Also, any appropriate or supported finite element analysis program, such as IFAD or MSC/NASTRAN, can carry out the analysis.

This technique allows engineers to generate more accurate results because approximation of input between analysis disciplines is minimized.

In summary, the process outlined in this paper shortens the time required to perform analysis. It also reduces error in communication of design geometry and errors in communication of analysis input. Reducing these errors allows greater consistency across analysis disciplines. The whole step-by-step process provides a means to store *design history* which is proving to be very important in todays mechanical engineering industry. Automation of the tedious and error prone tasks involved promotes *design through understanding*[6], and provides a structured design environment for young or inexperienced engineers. This contrasts with the environment of the past where design was done through tradition or experience. All in all, by letting the computer transfer these kinds of loading conditions, the engineer can *"focus on the design"* and not the tedium of the task at hand.

4 ACKNOWLEDGEMENTS

The author would like to thank and acknowledge the efforts of Michael Collingridge, Mitchell Walsh, and Bradley Wilkin of the Schlumberger Technologies, CAD/CAM Division. Their efforts have contributed to making this paper possible.

MSC/NASTRAN is a registered trademark of The MacNeal-Schwendler Corporation.

ADAMS and DRAM are registered trademarks of Mechanical Dynamics, Incorporated.

Bravo3 is a registered trademark of Schlumberger Technologies, Inc. Also, MECHANISMS, GRAFEM, IFAD, and SOLIDS MODELER are trademarks of Schlumberger Technologies, Inc.

[6]Design through understanding advocates using software analysis tools to build a body of knowledge about the problem and seek solutions (possibly analytical) via the resultant understanding.