ISPAN
INTERACTIVE STIFFENED PANEL ANALYSIS
A TOOL FOR
QUICK CONCEPT EVALUATION AND DESIGN TRADE STUDIES

John W. Hairr, William J. Dorris, J. Edward Ingram, and Bharat M. Shah*

Lockheed Aeronautical Systems Company

Marietta, Georgia 30063

523 –24

5/423

ABSTRACT

Interactive Stiffened Panel Analysis (ISPAN) modules, written in FORTRAN, were developed to provide an easy to use tool for creating finite element models of composite material stiffened panels. The modules allow the user to interactively construct, solve and post-process finite element models of four general types of structural panel configurations using only the panel dimensions and properties as input data. Linear, buckling and post-buckling solution capability is provided. This interactive input allows rapid model generation and solution by non finite element users. The results of a parametric study of a blade stiffened panel are presented to demonstrate the usefulness of the ISPAN modules. Also, a non-linear analysis of a test panel was conducted and the results compared to measured data and previous correlation analysis.

INTRODUCTION

Stiffened panel analysis computer programs have been in existence for many years. Typically, they are based on a combination of closed form solutions and a large data base of metallic stiffened panel test results. Developing a similar analysis capability for composite materials would be a formidable task due to the the lack of a complete composite stiffened panel test data base and due to the complex failure modes resulting from material anisotropy and variations in design/fabrication techniques. Closed-form solutions for the complex failure modes and out-of-plane response characteristics of the composite materials

933

are being developed; but so far, have not demonstrated sufficient applicability and accuracy. As finite element programs, and in particular, the pre- and post-processing programs have matured, it has become apparent that a more practical alternative to the closed-form solutions would be a non-linear finite element-based program which is user friendly. That alternative is envisioned here as an interactive input module which functions as an interface between the structures engineer and the finite element program (see Figure 1). The modules developed jointly at Lockheed Missiles and Space Company (LMSC) and at Lockheed Aeronautical Systems Company (LASC) are referred to as ISPAN, for Interactive Stiffened Panel Analysis.

Four user input/interface modules were developed; one each for flat and curved stiffened panels, truss-core panels and geodesic stiffened panels. Each module prompts the user for panel dimensions, properties, loads, boundary conditions and solution type (linear, buckling or post-buckling). The data supplied are then used to automatically prepare the finite element model. No knowledge of the finite element program commands or data format is required of the user. A typical stiffened panel geometry and solution procedure can be generated in a few minutes; solution time will obviously depend on the computer. Once a panel problem has been defined, the data are saved to a file, so that any changes to geometry, mechanical properties or loads may be accomplished by updating this database file with the data only requiring the changes. This allows the data for successive analyses be created in a fraction of the time required for the initial problem definition. This operational efficiency is valuable in the trade study.

The functioning of ISPAN modules, as illustrated in Figure 2, is analogous to a command module. In addition to interactively developing DIAL compatible geometry and materials files from the user supplied data, it also establishes procedures to communicate with the DIAL pre-processor, the solver, and post-processor. Thus it allows the user a direct control over the overall problem solving.

The Lockheed developed DIAL finite element code was selected for use in ISPAN because of its versatile non-linear and collapse analyses capability. DIAL also has very powerful pre- and post-processors requiring a relatively few number of commands . While the model generation data needed for DIAL is very concise, it is not intuitively obvious to an analyst not already familiar with the DIAL code which commands and which options should be used. Thus, if the structure to be analyzed is confined to a certain type, such as a stiffened panel, DIAL lends itself well to an interface procedure such as ISPAN. The

DIAL code also features a capability called syntactic input, which is a FORTRAN-like language that allows variable names for model data, and subroutines which use these variables to construct model generation commands. Finally, DIAL maintains a database, which contains model data, output data (internal loads, stresses, deflections, etc) and information generated during a post-processing session.

PROGRAM CAPABILITIES AND LIMITATIONS

The ISPAN modules are presently capable of analyzing the panel types and general cross-sections shown in Figure 3. The following list describes the options available to the user in the four modules

Stiffener
- cross-section type
- number of stiffeners
- spacing (may be irregular)
- flange widths, thickness
- materials , ply orientation

Skin
- width, length and thickness
- materials , ply orientations

Loading
- axial end load, N_x (tension or compression)
- lateral load perpendicular to the stiffener direction) N_y
- in-plane edge shear, N_{xy}
- normal pressure

Boundary Conditions	- Each of the four panel edges may be specified separately as free, simply-supported or clamped.
	- Vertical translations and/or rotations may also be specified across the width of the panel to simulate the rib/frame supports.
Solutions	- linear static
	- bifurcation (buckling)
	- post-buckling (arc length method)

DIAL Shell Element

All components of the stiffened panel models are constructed using a modified thick shell element as shown in Figure 4. This element is an eight node, super-parametric, doubly-curved thick shell element. The element was modified by separating the membrane, bending and transverse shear responses, thus allowing the adequate representation of both the laminate and the sandwich type structures. Linear, parabolic and cubic versions of this element are available in DIAL. The parabolic and cubic shell elements are vastly superior to the linear elements in terms of accuracy and are generally recommended. The parabolic elements were selected for use in ISPAN to provide optimum accuracy for a given level of complexity. An example of a typical flat, hat-stiffened panel model generated by ISPAN is shown in Figure 5.

Although the stiffener elements are generally distinct from the skin elements, it should be noted that the stiffener attach flanges are assumed to be included in the skin. Therefore, elements in the stiffener attach flange region should be given thickness and orientations which reflect the sum of the flange and skin in this region as indicated by the shaded area in Figure 6.

The ribs (frames) are not modeled with structural elements in this analysis. However, they are represented by constraints on the degrees of freedom at the rib locations. The details of this are discussed in the sub-section on boundary conditions.

Loads

In-plane axial (N_x), lateral (N_y) and shear loads (N_{xy}, N_{yx}) as well as normal pressure (P) may be applied to the model. Figure 7 shows the definition and sign convention of these load components.

The axial loads on the panel ends (N_x) are applied in such a way as to produce uniform strain, (ε_x). That is, the total applied load is effectively distributed to the base panel and the stiffeners according to their relative stiffnesses. This is accomplished in DIAL by forcing a set of edge "slave" nodes to displace as the weighted average of the displacements of the two "master" nodes at the panel corners as shown in Figure 8. The weights used are inversely proportional to the distances of the slave nodes to the master nodes. This method of applying axial loads simulates the loading produced by the surrounding wing or fuselage structure as well as in most stiffened panel test set-ups.

Boundary Conditions

Each edge (boundary) of the panel may be independently specified to be free, simply-supported or clamped. The simple supports are simulated constraining only the out-of-plane (normal to panel) degree of freedom at each node. Clamped conditions are simulated by constraining both normal translations and rotations along the edge. In addition to the constraints used to simulate the structural support provided by the adjacent structure, kinematic constraints are specified to prevent rigid body motions as shown in Figure 9. These kinematic constraints are generated automatically by the ISPAN modules Under normal circumstances, the kinematic constraints will produce no internal loading in the panel. If a pressure load were inadvertently specified without the balancing normal or hoop loads on the panel edges, then the kinematic constraints would provide the balancing loads. This undesirable condition would become obvious to the user once the deflected shape and/or stress contours are examined.

The option exists in ISPAN modules to provide out-of-plane support at specified locations on the stiffened panel in such a way as to simulate a rib or circumferential frame. The support is modeled by constraining the out-of-plane displacement normal to the panel at user specified locations. The inherent assumption in this modeling technique is that the rib or frame has sufficient stiffness to enforce a node in the buckled mode shape of the panel. The rotations about the y-axis at the rib locations may also be constrained in order to simulate a very rigid support such as a main frame. In most applications (intermediate frames or ribs), constraining these rotations will be unconservative.

Solutions

Three solution procedures, linear static, bifurcation (initial buckling) and post-buckling, are presently available in ISPAN. The linear static solution is straightforward and needs no explanation.

The initial buckling solution is an analysis to determine the multiplier on the applied load (eigenvalue) at which any part of the structure first becomes unstable. No knockdown factors nor initial imperfections are accounted for in this bifurcation analysis.

A post-buckling solution allows the user to determine the ultimate load capability of a structure. Some flight vehicle structures are permitted to buckle locally at load levels below ultimate, provided they retain the capability to carry the ultimate load without complete collapse. This loading condition can be analyzed using the post-buckling solution in DIAL, which enables an investigation of the integrity of the structure after initial buckling, to establish the ultimate allowable load based on strain cutoff, strain interaction or general instability /total collapse. The DIAL program uses the Arc-Length method, described in reference 2, to perform the non-linear analysis. This method requires the user to select a target point in the form of either load factor or a maximum displacement at a certain location (grid point). Since strength, rather than displacement is the criterion most often used for structural design, the load factor option is recommended for this type of analysis.

The solution procedure for conducting the non-linear analysis performs an initial buckling calculation first, and uses the resulting load and mode shape as the starting point for the non-linear post-buckling analysis. ISPAN multiplies the first eigenvector by 2%

(.02) to establish the shape of the imperfect panel at the zero load state. This procedure not only facilitates the non-linear solution procedure, but provides a more realistic analysis of actual structures; a feature in this analysis that is normally handled by knock-down factors in other solution techniques.

ANALYSIS-TEST COMPARISON OF POST-BUCKLED RESPONSE

The ISPAN module was used to analyze a composite flat stiffened panel for which test and previous analysis work already exists (ref 1). The purpose of this analysis was to verify the accuracy of the post-buckled analysis solution in DIAL. The panel in this case is an "I" stiffened panel 81.3 cm long (parallel to stiffeners) and 61.0 cm wide with four stiffeners spaced at 17.8 cm. The panel was loaded in pure compression.

The model geometry generated by ISPAN is shown in Figure 10. As in all ISPAN modules, modified thick shell elements were used for all components. The deflected shape of the panel at a load of 92.7 kips is shown in Figure 11. This shape, which consists of five half-waves, is identical to that observed in the test panel and in the STAGS analysis of ref. 1.

The comparisons of the DIAL analysis with the results from ref. 1 in terms of load vs. end shortening and load vs. strain curves are presented in Figures 12 and 13 respectively, and show reasonably good agreement. ISPAN presently does not include the capability to generate tapered elements such as the attach flanges in the test panel. This difference is thought to be responsible for the disagreement between test results and the ISPAN-generated DIAL model results. Plans for future enhancements to the ISPAN program may include incorporation of the interface element, which can be used to simulate the bond between the stiffener and the skin panel. This interface element allows the determination of the longitudinal and transverse shear and normal stresses (and strains) at each load step in the non-linear solution; and thus can be used to predict stiffener interface failure such as the one observed in the ref. 1 test.

PARAMETRIC STUDY

A trade study was conducted to illustrate one of the uses of ISPAN. A 24 in. by 24 in. blade-stiffened panel was analyzed with varying stiffener heights and spacings. The loading for the study was uniform compression and the boundaries were simply supported. Figure 14 shows an example of one of the blade stiffened panel model geometries generated with ISPAN, in which the stiffener height was 2.0 in., and the spacing was 4.0 in. The combinations of height and spacing considered in this study are shown in Figure 15 along with the critical loads resulting from the linear and non-linear solutions. Both the applied load at the strain cutoff value and the critical or collapse load are included in Figure 15 These results are presented in graphical form in Figure 16, in which it may be seen that for a given load, there is an optimum combination of height and spacing. In this limited study, the skin and stiffener thickness and laminate descriptions were not allowed to vary; and obviously a comprehensive analysis to determine the optimum panel for a given application would involve not only variations in thickness and properties, but panel length, stiffener type and many other variables.

CONCLUSION

The ISPAN interface modules allow both experts and non-users of finite element codes access to a powerful non-linear analysis procedure to enhance their efficiency and analysis accuracy. The interface module procedure fills a void in the available methods of analysis for anisotropic stiffened panel analysis. The expected benefit from this program is the ability to consider a broad range of stiffened panel geometries and materials for multi-axial loads application during the trade study phase of a new design also accounting for the critical failure modes .

The analysis of an I-stiffened test panel shows that the accuracy of the ISPAN non-linear analysis procedure compares favorably with test data and with other finite element analyses.

Plans for future enhancements to ISPAN include incorporation of the existing DIAL skin-stiffener interface element into the automatic mesh generation . The interface element

is used to represent the stiffness of the bond between the stiffener (s) and the skin. The output stresses and strains for the interface element will allow bond failure modes to be included in the analysis.

ACKNOWLEDGMENT

The authors wish to thank Messrs. Wayne Brown, Tony Wei and Dr. Norm Cyr, of Lockheed Missiles and Space Company for their work in the development of the ISPAN program(s). The authors also thank Dr. Randall Davis (NASA/LaRC) for his review and comments.

REFERENCES

1. Starnes, James H., Jr., Knight, Norman F., Jr., and Rouse, Marshall "Postbuckling Behavior of Selected Flat Stiffened Graphite-Epoxy Panels Loaded in Compression," AIAA Paper No. 81-0543, 1982.

2. Anon., "DIAL Structural Analysis System User's Manual", Version L3D3, August 1,1991, Lockheed Missiles and Space Company, Sunnyvale California.
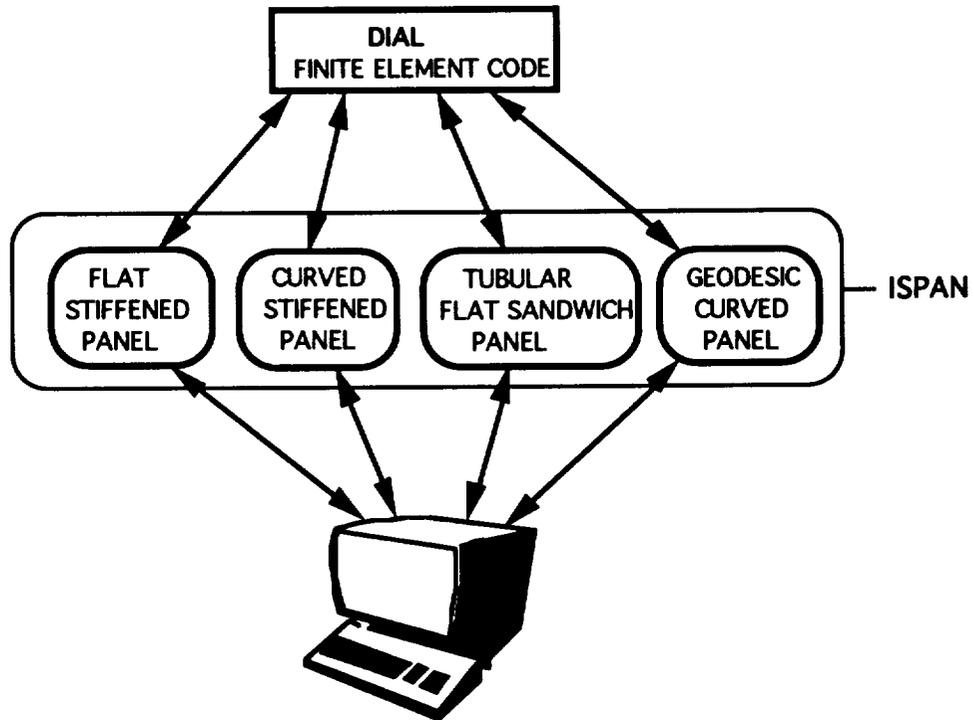
Figure 1 ISPAN - The Interface Between the Non-Finite Element Users and DIAL
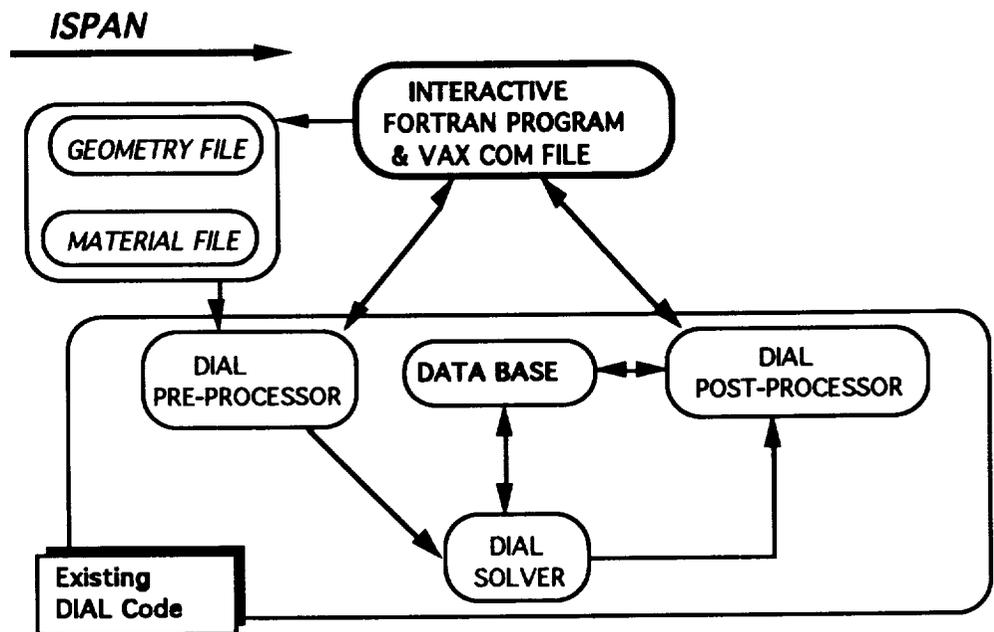
Figure 2 ISPAN Module Functional Flow

942

MODULES 1 & 2: → (FLAT AND CURVED STIFFENED PANELS)

MODULE 3: → (TRUSS CORE)

MODULE 4: → (GEODESIC)
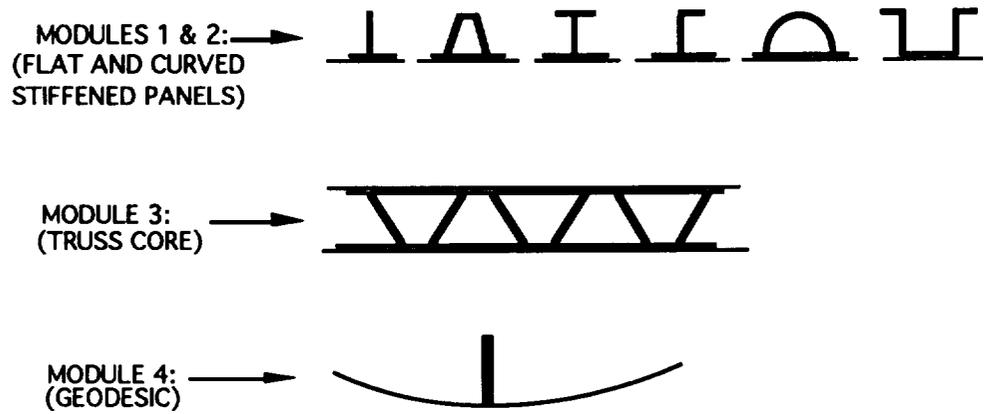
Figure 3 Stiffener Cross Sections Available in ISPAN

- SUPER-PARAMETRIC, DOUBLY CURVED THICK SHELL ELEMENT

- MEMBRANE, BENDING AND SHEAR RESPONSES ARE SEPARATED (TO SIMULATE LAMINATE CONSTRUCTION)
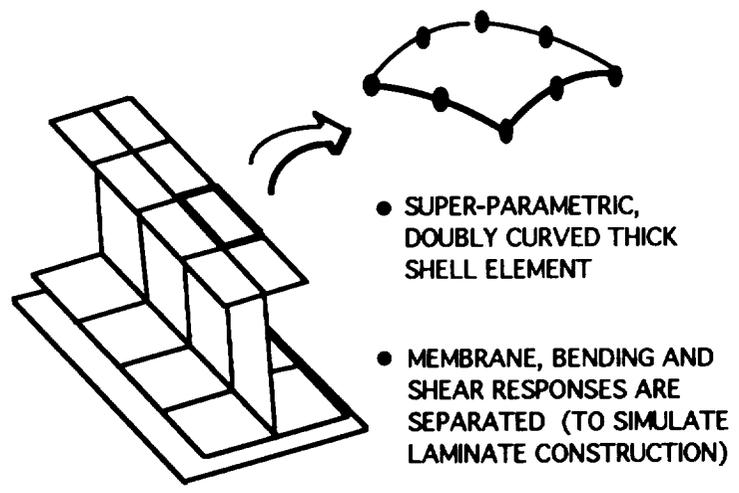
Figure 4 Shell Element Type Used in ISPAN

943
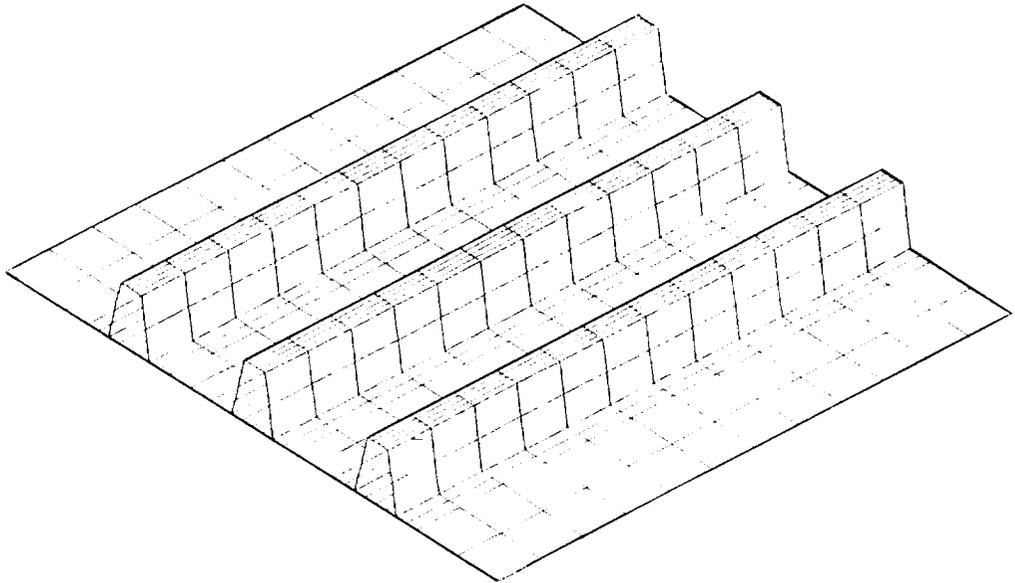
Figure 5 Hat Stiffened Panel Model Generated by ISPAN

TOTAL THICKNESS MODELED
AS ONE SHELL ELEMENT
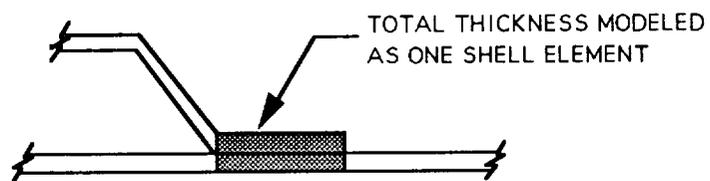
Figure 6 Attach Flanges of Stiffeners and Skin Elements are Modeled
as One Shell Element

944
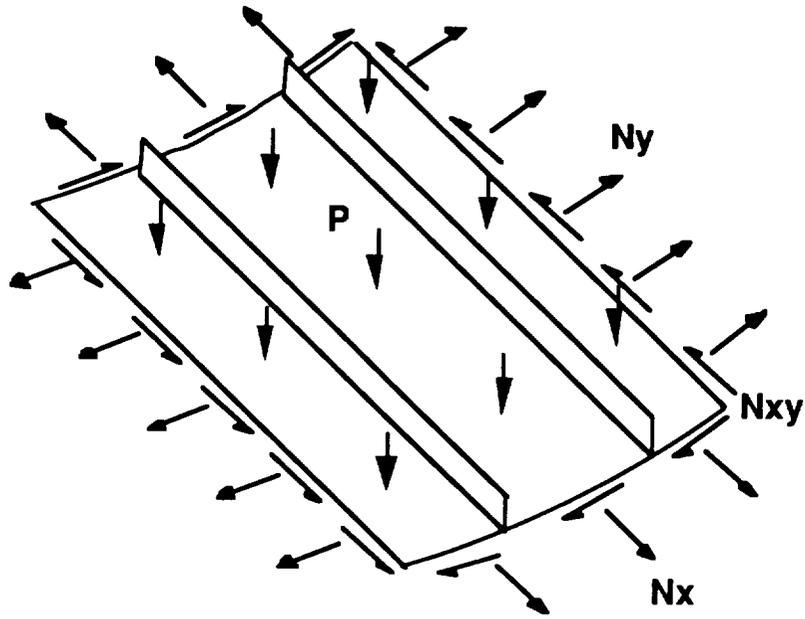
Figure 7 Applied Loads Definition
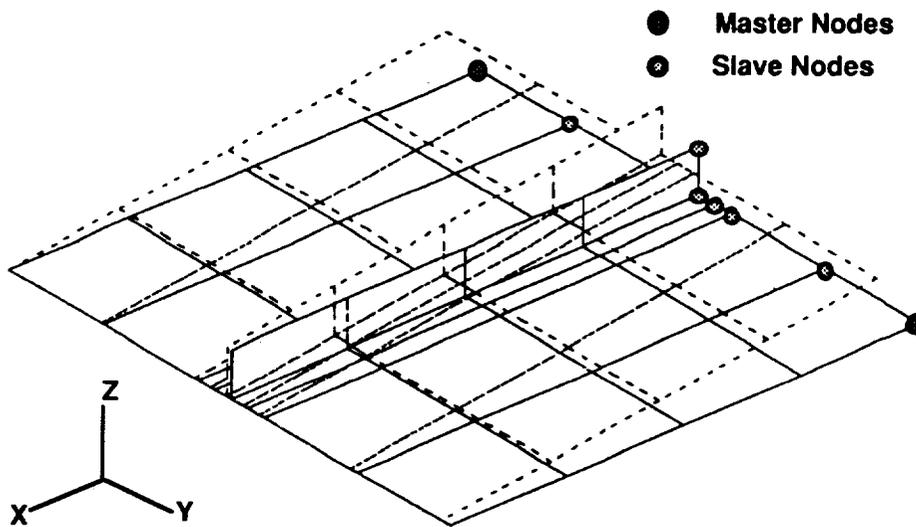
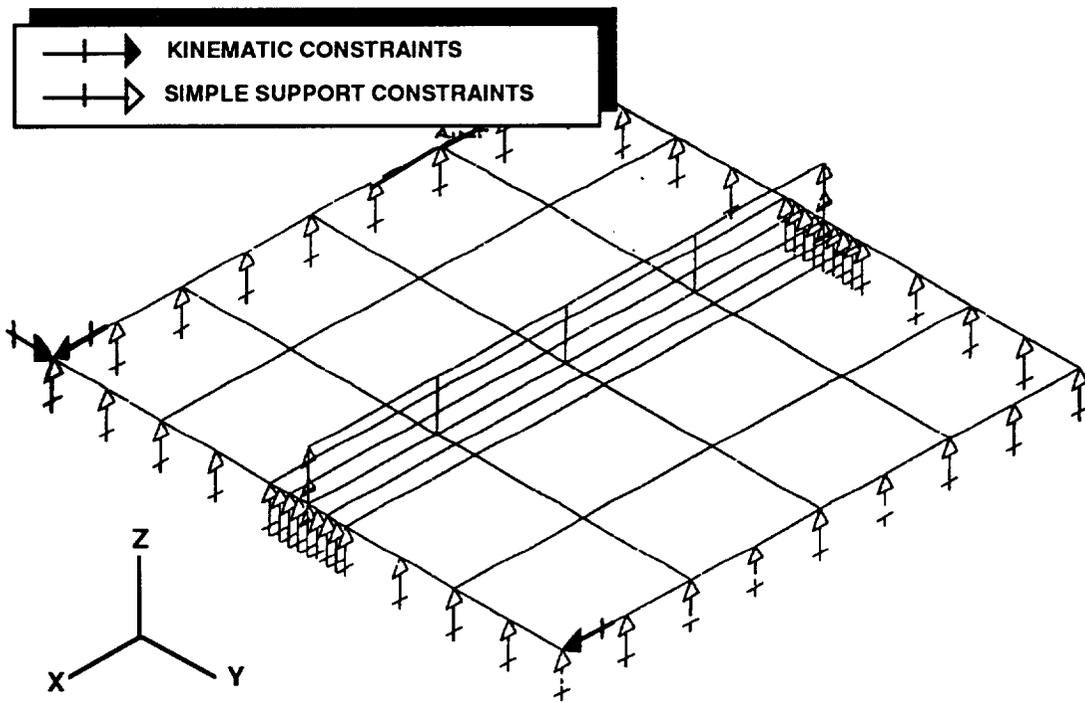Figure 8 Master and Slave Nodes in CONODE Command

KINEMATIC CONSTRAINTS

SIMPLE SUPPORT CONSTRAINTS

Figure 9 Typical Constraint Vectors for Flat Panel
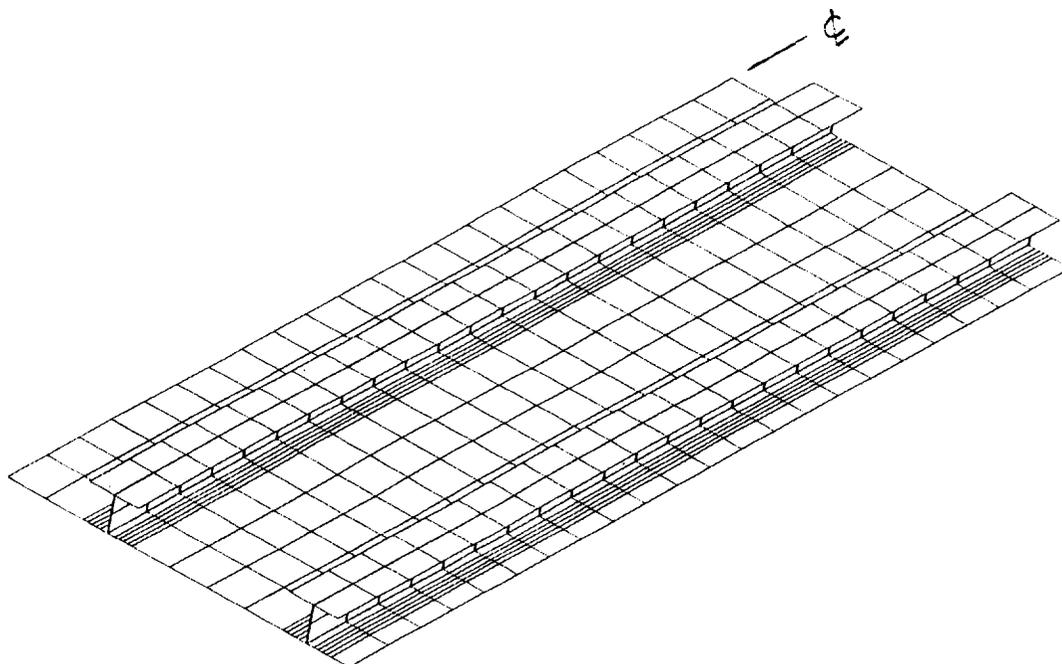
Figure 10 DIAL Model Geometry of Test Panel From Reference 1.

946

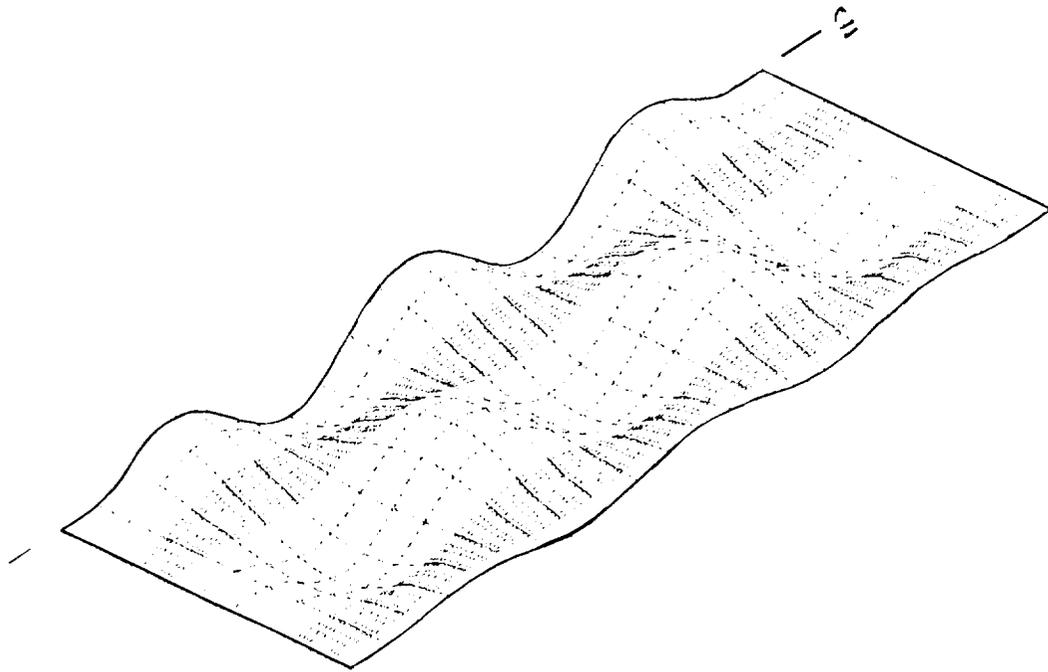Figure 11 Deflected Shape of DIAL Test Panel Model at Final Load Step

Load (lbs)

■ Starnes, et al.
O DIAL Nonlinear Soln
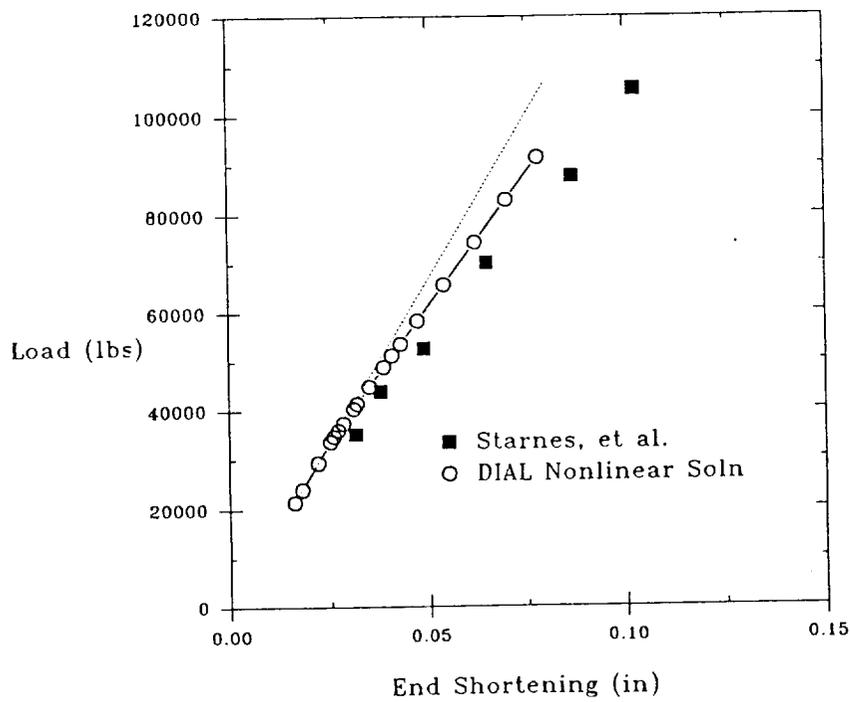
End Shortening (in)

Figure 12 Comparison of Load-Displacement Results from DIAL and from Ref. 1 Test and Analysis
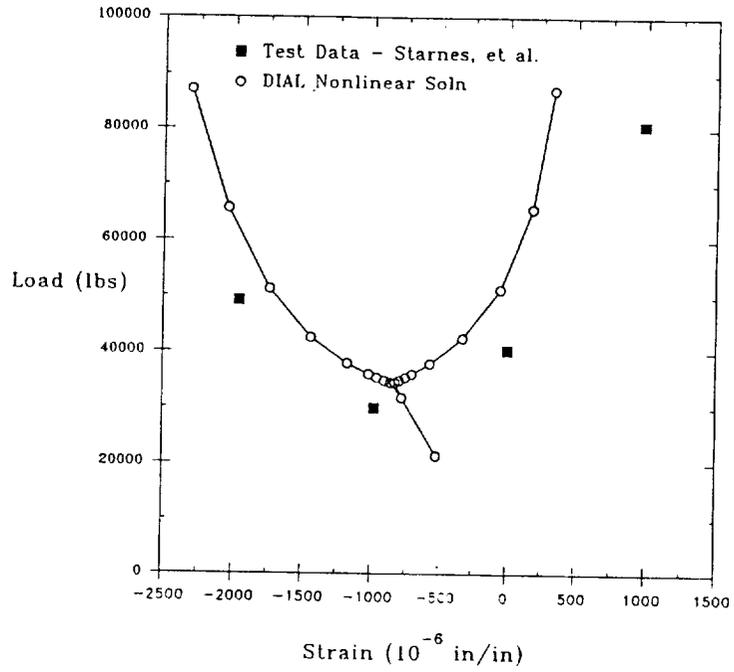
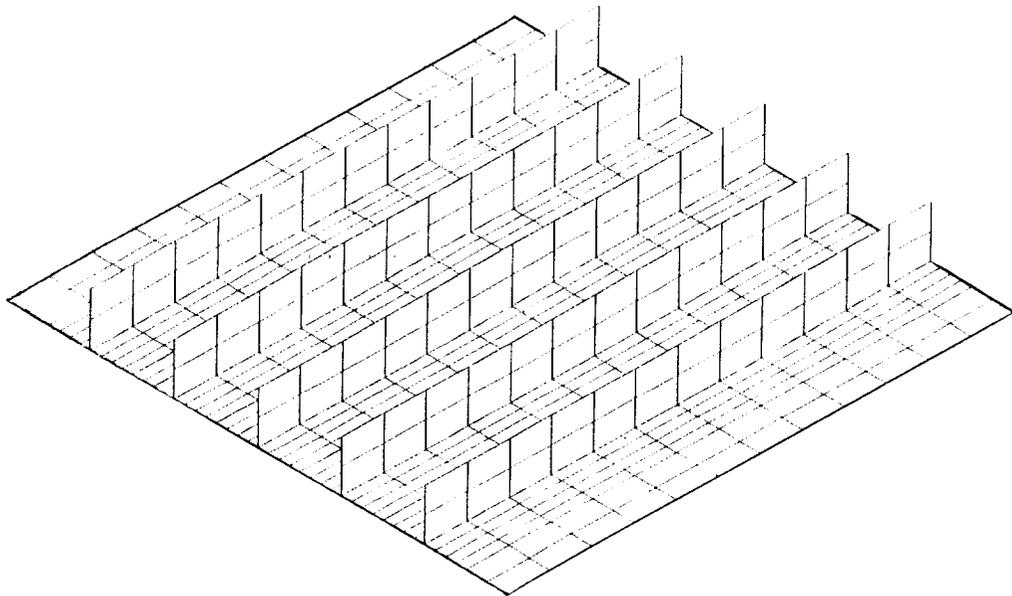Figure 13 Comparison of Load-Strain Results from DIAL with Test Data (ref. 1)

Figure 14 Typical Blade Stiffened Panel Model Geometry from Parametric Study

Number of Stiffeners	SPACING b (in)	HEIGHT h (in)	t bar (in)	N_x co (lb/in)	N_x cr (lb/in)
5	4.0	0.75	.0988	-3019.8	-2789.4
5	4.0	1.00	.1050	-3357.7	-3340.5
5	4.0	1.50	.1175	-4033.3	-5475.0
5	4.0	2.00	.1300	-4708.9	-3897.9
5	4.0	2.25	.1362	-5046.7	-3871.8
3	6.0	0.75	.0913	-2614.5	-1616.7
3	6.0	1.00	.0950	-2817.2	-2175.0
3	6.0	1.50	.1025	-3222.5	-3015.3
3	6.0	1.75	.1063	-3425.2	-3174.0
3	6.0	2.25	.1138	-3830.6	-2417.1
2	8.0	1.50	.0950	-2817.2	-1146.7
2	8.0	2.00	.1000	-3087.4	-2827.1
2	8.0	2.25	.1025	-3222.5	-2982.3

Figure 15 Stiffened Panel Parametric Study Dimensions And Results
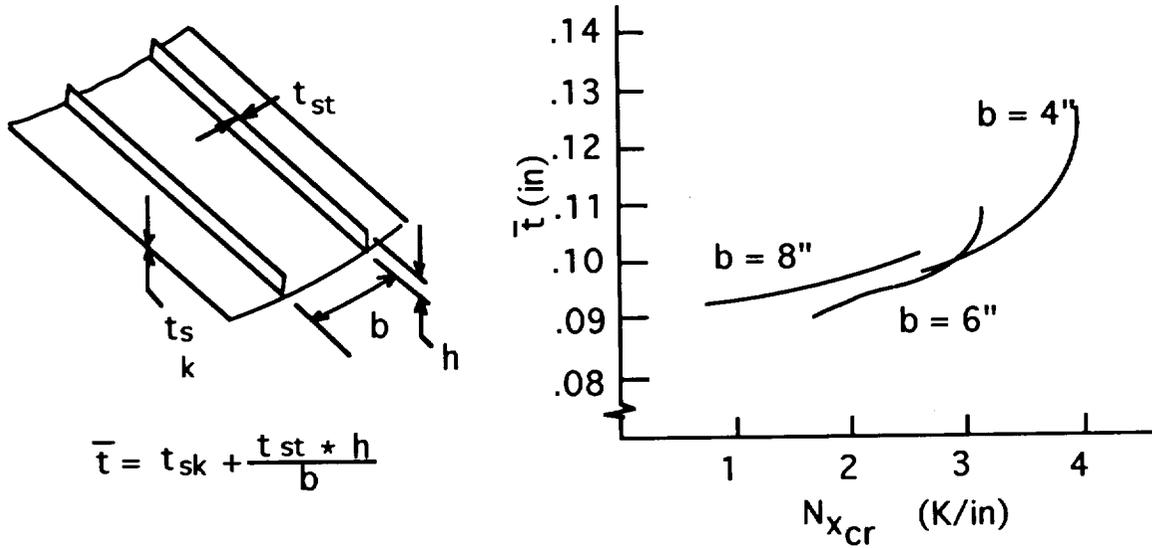

$$\bar{t} = t_{sk} + \frac{t_{st} * h}{b}$$

Figure 16 Results of Parametric Analysis of Blade Stiffened Panel