GRID SENSITIVITY ANALYSIS USING MSC/NASTRAN

Gopichand Somayajula
Ford Motor Company, Dearborn, Michigan
and
James E. Bernard
Iowa State University, Ames, Iowa

Abstract

This paper presents a method to compute the sensitivities of displacements and natural frequencies to the changes in grid locations. The sensitivities are computed within MSC/NASTRAN using the semi-analytical method via a DMAP program. New software was developed to assist in the selection of design variables, and plot the spatial variation of the sensitivities on the finite element(FE) model using PDA/PATRAN. Two example problems demonstrate the technique.

1 INTRODUCTION

Structural design is a complex process which has the goal of generating a product which satisfies performance requirements in a cost-effective manner. Traditionally, the design methodology relies on analyzing the design to determine its structural response and then repeatedly remodifying and reanalyzing to obtain more desirable performance. The remodification and reanalysis can either be an intutively based optimization under the control of the designer or a mathematically based optimization under computer control.

Sensitivity methods have facilitated the optimization process by replacing the solution of a finite element problem with an easy-to-compute approximation. MSC/NASTRAN implemented this capability for changes in element related properties such as cross-sectional areas and material properties, but not, so far, for general changes in geometry which we characterize here as grid changes[1].

The purpose of this paper is to develop and implement the sensitivity method for solid element structures using grid locations as the design variables inside MSC/NASTRAN. Using this approach, effects of moving the surface of a specific portion of the solid element structure on a structural response can be determined without constructing new finite element models. Specification of the design variables, which include both the grid labels and the associated direction vector for grid movement, is not a trivial task. Software is presented which assists the specification of design variables and the display of the sensitivity results and the associated direction vectors on the finite element model.

2 GRID SENSITIVITIES

Grid sensitivity analysis is the calculation of changes in the response of a structure due to changing the finite element grid point locations. There are two basic approaches to the calculation of sensitivity derivatives. The first approach, known as implicit differentiation, is based on differentiation of the discretized finite element system[2]. The other, which is based on the variation of continuum equations, is known as the variational or material derivative approach[3]. The method used in this paper is based on the implicit differentiation. The basic theory underlying the displacement and natural frequency sensitivity analyses is briefly explained in the follwing sections.

2.1 Displacement sensitivities

The governing equilibrium equation for a structure subject to quasistatic load is:

$$\mathbf{K}(\mathbf{v})\mathbf{x} = \mathbf{f}(\mathbf{v}) \tag{1}$$

where $\mathbf{v}$ is the shape design variable vector of order s, $\mathbf{K}(\mathbf{v})$ is the symmetric non-singular structural stiffness matrix of order n, $\mathbf{f}(\mathbf{v})$ is equivalent nodal load vector of order n, x is nodal displacement vector of order n. The design variables are movements of grid point locations along the chosen directions. The order n corresponds to the degrees of freedom of the finite element model and order s refers to the number of design variables in the model.

Differentiation of equation (1) with respect to the design variables v_i yields:

$$\mathbf{K}\frac{d\mathbf{x}}{dv_i} = \left[\frac{\partial \mathbf{f}}{\partial v_i} - \frac{\partial \mathbf{K}}{\partial v_i}\mathbf{x}\right] \tag{2}$$

The right hand side of the equation is called pseudo-load vector since the equation(2) resembles equation(1). In the absence of body forces, the applied load $\mathbf{f}$ is usually independent of the design variables. Thus we assume $\frac{\partial \mathbf{f}}{\partial v_i}$ to be zero. The remaining quantity on the right hand side of equation(2) is obtained by multiplying $\frac{\partial \mathbf{K}}{\partial v_i}$ by the displacement vector $\mathbf{x}$. Equation (2) can be solved by the same solution algorithm used for equation (1), taking advantage of the fact that $\mathbf{K}$ is available in factored form from the solution of equation (1). The number of pseudo-load vectors used in the sensitivity calculation is $s \times t$, where s is the number of design variables and t the number of applied load cases.

Equation (2) indicates that computation of the gradients $\frac{d\mathbf{x}}{dv_i}$ requires evaluation of gradients of the stiffness matrix and load vector with respect to the design variable v_i. In some special cases such as bar and memberane structures, it is possible to seperate element stiffness matrices as products of constant matrices and the parameter with respect to which the derivative is taken and makes the derivative computations simple. But, a general formulation is not available for derivatives with respect to the shape variables. Finite difference methods provide a practical approach to determine shape derivatives of the stiffness matrix.

The finite difference is introduced by perturbing each design variable by a small amount and regenerating the global stifness matrix which is then used in a finite diffrence equation to determine the derivative of the global stiffness matrix. For example, the first derivative of a stiffness matrix $\mathbf{K}(v_i)$ with respect to a design variable v_i can be approximated with the central difference method as:

$$\frac{d\mathbf{K}}{dv_i} = \left[\frac{\mathbf{K}(v_i + \epsilon) - \mathbf{K}(v_i - \epsilon)}{2\epsilon}\right] \tag{3}$$

Where ϵ is a small perturbation of the chosen grid location along the specified direction. The choice of the magnitude of the perturbation is crucial for the accuarcy of the derivative. A large value of ϵ produces large truncation errors, while a small value of ϵ may result in large round-off errors. Careful selection of ϵ allows the use of forward finite difference approximation which requires less computational effort as compared to the central diffrence approximation. Liefooghe et al.[4] investigated the influence of perturbation size ϵ on the accuracy of the sensitivities.

The implementation of finite difference techniques is often much simpler than the analytical techniques for obtaining the derivatives and largely independent of the type of finite elements used in the analysis. The method is particularly useful when the analysis program source code is not available, which is the case in the commercial codes. The sensitivity calculation using finite difference derivatives is often referred to as a "semi-analytical" method[5] or a quasi-analytical method[6].

2.2 Natural frequency sensitivities

Governing equation for the free vibration response of a structure is often cast in the form

$$\mathbf{K}(\mathbf{v})\phi_i = \lambda_i \mathbf{M}(\mathbf{v})\phi_i \tag{4}$$

where $\mathbf{M}(\mathbf{v})$ is the symmetric structural mass matrix of order n, λ_i is the eigenvalue related to the natural frequency, and ϕ_i is the associated eigenvector. Assuming that the mode shapes are normalized with respect to the mass matrix $\mathbf{M}$,

$$\phi_i^T \mathbf{M} \phi_i = 1 \tag{5}$$

$$\phi_i^T \mathbf{M} \phi_j = 0 \tag{6}$$

We can diffrentiate equation (4) with respect to design variable v_i to get

$$\frac{\partial \mathbf{K}}{\partial v_i}\phi_i + \mathbf{K}\frac{\partial \phi_i}{\partial v_i} - \frac{d\lambda}{dv_i}\mathbf{M}\phi_i - \lambda_i \frac{\partial \mathbf{M}}{\partial v_i}\phi_i - \lambda_i \mathbf{M}\frac{\partial \phi_i}{\partial v_i} = 0 \tag{7}$$

After premultiplying by ϕ_j^T and substituting equation (5) and (6), Equation (7) can be simplified to:

$$\frac{d\lambda}{dv_i} = \phi_i^T \left[\frac{\partial \mathbf{K}}{\partial v_i} - \lambda_i \frac{\partial \mathbf{M}}{\partial v_i} \right] \phi_i \tag{8}$$

$\frac{\partial \mathbf{K}}{\partial v_i}$ and $\frac{\partial \mathbf{M}}{\partial v_i}$ can again be evaluated using finite difference methods.

3 DESIGN MODEL

Preparation of the model, including the definition of the design variables, is the crucial step in the sensitivity analysis. In grid sensitivity analysis, the design variables are the locations of the FE nodes on the surface of the structure. Since the sensitivity calculation invoves the differentiation of the structural matrices **K** and **M**, the computation burden of the method grows with the number of design variables. Further, the design changes initially envisioned by the designer are in terms of changes in components, not in grid locations, thus it is desirable to cast the design changes in terms of one key measure, for example a radius, rather than multiple measures associated with grid locations. Therefore it is useful to create a design model with only a few design variables which describe the potential design changes.

To this effect, the concept of *control node* is introduced. Control nodes reduce the number of design variables and allow the use of meaningful design parameters as design variables. For example, with proper geometric linking of FE nodes to carefully chosen control nodes, the thickness of a flange, the radius of a shaft, or the angle of a groove can be used as design variables. Typical description of the design model then must include the decomposition of the structure into a few regions of simple geometry called *design elements* which define geometric linking between the movement of the regions and the associated control nodes. The movements of the control nodes are then thought of as the design variables. Thus the control nodes, along with all their associated nodes, are perturbed to obtain the derivatives of the global stiffness and mass matrices during the sensitivity analysis

3.1 GEOMETRIC LINKING

Geometric linking basically expresses the movement vector for the FE nodes of a design element as a function of the control node movement, and the linking equation depends upon the geometry of the design element. The linking procedure is illustrated in Figure 1 for three common design element types- cylindrical, angular, and parallel elements. Each element is associated with a control node(design variable). Radius 'ρ' of the cylindrical surface is the control node for the cylidrical element. The slope 'θ' of the angular surface is the control node for the angular element. The thickness 't' of the panel is the control node for the parallel element. The movement vectors $\vec{d}$ for the FE nodes due to a perturbation ϵ in the control node are shown for the three cases in the Figure 1. The control nodes can be perturbed to obtain the derivatives of the global mass and stiffness matrices with respect to the design variables.

3.2 DESIGN VARIABLE LINKING

Often it is unnecessary or even undesirable that each design element be defined by unique design variables. Design variable linking combines two or more design variables into single independent design variable. Thus a single control node movement controls the movement of all the dependent design elements. This feature is quite useful for constructing meaningful design variables. For example, in Figure 2, the thickness of a panel is made a design variable by linking the two parallel design elements on both the sides. A change in the design variable causes the movement of the parallel elements away/towards from each other.

3.3 SOFTWARE

Software has been developed to assist the model preparation and sensitivity calculation/display tasks. The software altogether consists of 5 modules which opearte interactively. The different modules of the software will be discussed briefly here.

3.3.1 VARNODE

The VARNODE module consists of subprograms used to identify the nodes associated with various design elements. The surface regions for the design elements can be either from the finite element model or from the surface patches defined during the finite element mesh generation using PDA/PATRAN(a commercial finite element pre-/post-processor from PDA Engineering).

3.3.2 GLINK

The GLINK module establishes the geometric linking between the design element FE nodes and the associated control node.

3.3.3 DLINK

The DLINK module provides design variable linking feature. It combines the two or more design elements to form a single design variable as per the user's directives.

3.3.4 PERTURB

The PERTURB module generates the NASTRAN data files due to perturbation in the control nodes. These data files are used to generate perturbed global stiffness/mass matrices which are used to compute the derivatives.

3.3.5 PATGRSEN

The PATGRSEN module generates PATRAN results files from the grid sensitivity data and design element nodes list for displaying the color-coded plots of the structure showing the sensitivities of the various design elements. Different colors indicate different sensitivity values. This module also generates results files to display the FE node movement vectors on the finite element model.

4 IMPLEMENTATION

Using the concepts described in the previous sections, the overall sensitivity analysis task may be segemented into the following steps.

1. Finite element analysis of the design for structural response- displacements, natural frequencies etc.
2. Preparation of design model which includes the design variable definition v_i.
3. Computation of $\frac{\partial \mathbf{K}}{\partial v_i}$ and $\frac{\partial \mathbf{K}}{\partial v_i}$ using finite difference method.
4. Computation of displacement and natural frequency sensitivities.

MSC/NASTRAN is used to determine the structural response, and generate the structural matrices for perturbed design variables. The sensitivity computation method is incorporated into MSC/NASTRAN using DMAP programs. Two independent DMAP programs are developed to compute displacement and natural frequency sensitivities. The computation procedure requires three types of NASTRAN runs:

- Analysis run: Determines the structural response- displacements, natural frequencies, mode shapes, and saves necessary data blocks/matrices for further use in sensitivity analysis.
- Matrix creation runs: Generate/save the structural matrices for perturbations in the design variables. Number of runs depend upon the number of design variables.
- Sensitivity run: Generates the sensitivities using the independent DMAP procedure.

NASTRAN's solution 61 is used for determining displacements due to static loads and solution 63 for the natural frequencies and mode shapes. DMAP alters are inserted into these solutions to save the necessary data blocks and matrices for use in sensitivity calculations. In the matrix creation runs, similar alters are used with these solutions to generate/save structural matrices for the perturbed design variables. New matrix generation runs are needed for each design variable. The efficiency of these runs can be improved by using the information created in the first run i.e. USET table, GM table, EQEXIN table etc. With this information, some operations can be bypassed when formulating the new structural matrices for other design variable perturbations.

Figure 3 contains the DMAP statements necessary to implement the displacement sensitivity calculations. The structural stiffness matrix, displacement solution vector, and the other necessry data blocks from the analysis run are retrieved using DBFETCH module. The stiffness matrices for the perturbed design variables from the matrix creation runs are retreived using INPUTT2 modules. The finite difference derivative of the stiffness matrix with respect to design variable is calculated using ADD modules. The perturbation ϵ is specified through a parameter card in bulk data. Finally the displacement sensitivity is computed using MATMOD, MPYAD, FBS, SDR1, and SDR2 modules. The sensitivity results OUGV1 are printed using OFP module. This process is implemented in two nested do loops. The inner loop LOOPSUB repeats as many times as the number of applied load cases. The outer loop LOOPVAR repeats as many times as the number of design variables. The number of repetitions for these two nested loops are set on the respective REPT modules.

Figure 4 contains the DMAP statements necessary for the natural frequency sensitivity calculations. The strategy is similar to the displacement sensitivity calculation. The structural mass and stiffness matrices, eigenvalue table, and eigenvector matrix from the analysis run are retrieved using DBFETCH module. The pertubed structural matrices from the matrix creation runs are retrieved for a given design variable again using INPUTT2 modules. Next, the derivatives of the mass and stiffness matrices are computed using ADD modules. Finally, the sensitivity is computed using ADD and SMPYAD modules, and the result DLAM is printed. The process is repeated using LOOPVAR for all the design variables.

5 NUMERICAL EXAMPLES

Two example problems are presented which illustrate the capabilities of the grid sensitivity computation methodology. The first is the eigenvalue sensitivity analysis for a L-shaped bracket. The second example is the

Table 1: Eigenvalue sensitivities to changes in bracket thickness.

Design variable	Sensitivity
Variable 1	2.024
Variable 2	1.848
Variable 3	0.920
Variable 4	-0.036
Variable 5	-0.398
Variable 6	-0.500

displacement sensitivity analysis for a crankshaft of an automotive engine.

5.1 L-shaped Bracket

A L-shaped bracket with one end fixed is considered here to demonstrate the eigenvalue sensitivity capability. The bracket is made of phenolics, and the dimensions are shown in Figure 5. The bracket is modelled using 64 nodes and 21 CHEXA elements. Figure 6 presents the finite element model. The grid points on the upper edge of the bracket are constrained in all degrees-of-freedom to model the fixed end condition. The model was analyzed for eigenvalues and eigenvectors, and its fundamental eigenvalue is 1.36346E+09(5877 Hz).

This example concerns the calculation of the sensitivity of this first eigenvalue to the thickness of the bracket. To formulate the thickness design variables 't_i', the top surface of the bracket was divided into six parallel design elements(Figure 7) with the control node movement direction normal to the top surface. Geometric linking feature was used to link the control node movement to the FE nodes. A perturbation of $0.01t_i$ was used to compute the finite difference derivatives of mass and stiffness matrices. The sensitiviies were computed for the six design variables, and presented as ratio of % change in eigenvalue to % change in design variable in Table 1. The grids in the vicinity of the fixed end have the highest sensitivity and those at the free end have the lowest sensitivity.

Table 2: Displacement sensitivities to changes in crankshaft dimensions.

Design variable	Sensitivity
Cheek thickness C1	0.014
Cheek thickness C2	0.696
Cheek thickness C3	1.057
Crankpin diameter P1	2.168

5.2 Crankshaft

Lateral stiffness is an important aspect of the crankshaft design. In this example, the displacement sensitivity capability is used to determine the lateral bending deflection sensitivity due to an end load of 10,000 lb. The finite element model(Figure 8) has 9790 nodes and 8475 solid elements; the material is nodular iron. The model was analyzed for displacements, and the lateral displacement at the load point was found to be 8.618E-3 inches.

In this example we determine the lateral displacement sensitivity for the load point due to changes in cheek thicknesses and crankpin diameter. The design variables, cheek thicknesses and crankpin diameter, were constructed using the geometric linking and design variable linking features. The cheek thicknesses were formulated using the parallel design elements with control node movement direction parallel to the crankshat axis of rotation, and the crankpin diamter variable was formulated using a cylindrical design element. All the crankpins were constrained to have a single diameter value. All the pin diameter variables were combined into a single design variable using design variable linking. A perturbation of $0.01v_i$ was used to compute the finite difference derivatives. The construction of cheek thickness and pin diameter variables are shown in Figure 9 and Figure 10 respectively. The sensitivities were computed for the four design variables. Table 2 presents the ratio of % change in lateral displacement to % change in design variable.

6 CONCLUDING REMARKS

This paper describes a method of obtaining the shape design sensitivity information using MSC/NASTRAN. Geometric linking and design variable linking concepts provided the capability to construct meaningful design variables out of finite element nodes. Software was presented to assist the design variable formulation, and a DMAP program is developed to compute the sensitivities. The utility of the methodology was illustrated through examples.

ACKNOWLEDGEMENTS

The authors thank D. H. Yun. and R. G. Delosh of P/T NVH Design Analysis for their support and encouragement. Special thanks to C. D. Kennel for his help in providing the crankshaft FE model, and Kalyani Somayajula for her help in preparing this paper.

References

[1] D.V. Wallerstein, Design enhancement tools in MSC/NASTRAN. *Recent Experiences in Multidisciplinary Analysis and Optimization*, NASA CP-2327, 505-526(1984).

[2] M.E. Botkin, Shape optimization of plate and shell structures. AIAA Journal **20**, 268-273(1982).

[3] K.K. Choi and E.J. Haug, Shape design sensitivity analysis of elastic structures. J. Struct. Mech. **11**, 231-269(1983).

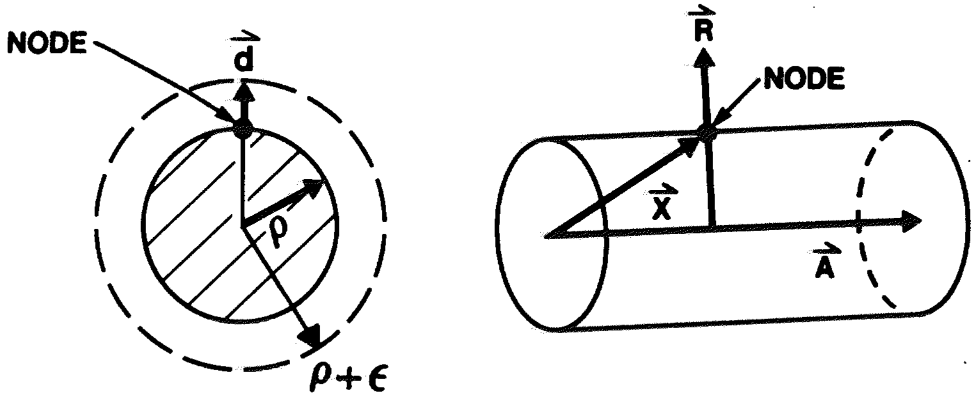
[4] D. Liefooghe, Y.K. Shyy, and C. Fleury, Shape sensitivity analysis using low and high order finite elements. AIAA paper 88-2431, Proceedings, AIAA/ASME/ASCE/AHS 29th Structures, Structural Dynamics and Materials Conference, Williamsburg, VA, April 18-20, Part 3, 1656-1666.

[5] J.A. Bennet, Structural optimization in the industrual design process. Int. J. Veh. Des. **5**(1 & 2), 115-128(1984).

[6] Cheng Gengdong and Liu Yingwei, A new computation scheme for sensitivity analysis. Eng. Opt. **12**, 219-234(1987).

GEOMETRIC LINKING

A. CYLINDRICAL DESIGN ELEMENT

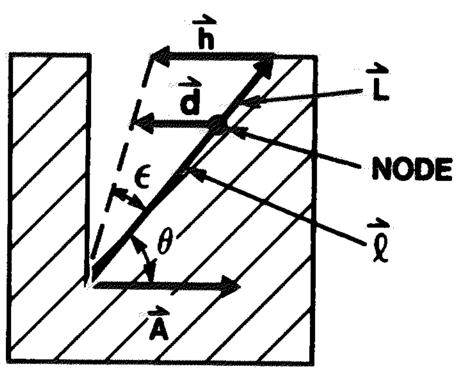


$$\vec{R} = [I - \vec{A}\vec{A}^T]\,\vec{X}$$

$$\vec{r} = \frac{\vec{R}}{\|\vec{R}\|}$$

$$\vec{d} = \epsilon\,\vec{r}$$

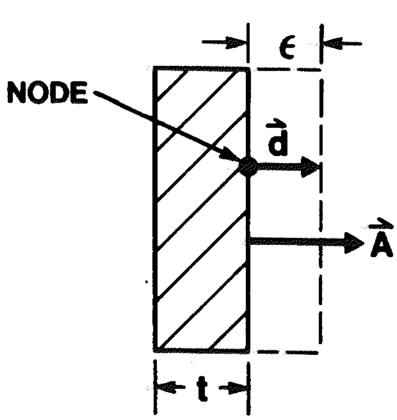
B. ANGULAR DESIGN ELEMENT



$$\|\vec{h}\| = \frac{\|\vec{L}\| \sin \epsilon}{\sin(\pi-\theta-\epsilon)}$$

$$\vec{d} = -\frac{\|\vec{h}\|\;\|\vec{\ell}\|}{\|\vec{L}\|}\;\frac{\vec{A}}{\|\vec{A}\|}$$

C. PARALLEL DESIGN ELEMENT



$$\vec{d} = \frac{\epsilon\,\vec{A}}{\|\vec{A}\|}$$

$\vec{d}$: NODE MOVEMENT VECTOR DUE TO CHANGE ϵ IN CONTROL NODE.

FIGURE: 1

TWO PARALLEL DESIGN ELEMENTS ARE COMBINED TO FORM A THICKNESS DESIGN VARIABLE

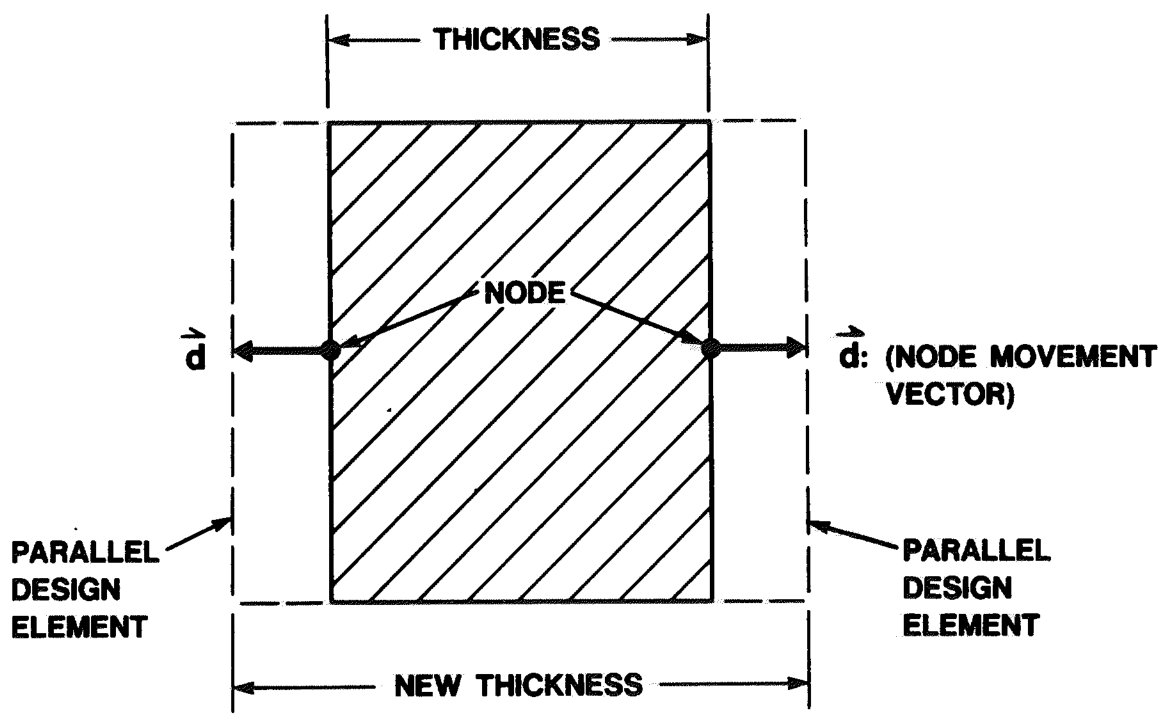


FIGURE: 2

MSC/NASTRAN IMPLEMENTATION OF DISPLACEMENT SENSITIVITY ANALYSIS

(VERSION 65)

```
BEGIN $
PARAM //NOP/V,N,ON1=1  $
PARAM //NOP/V,N,OP2UN=15 $
DBFETCH /LLL,KLL,ULV,KFS,KSS/V,Y,SOLID///DBSET2 $
DBFETCH /USET,CASEDR,EQEXINS,,/V,Y,SOLID///DBSET2 $
PARAM //NOP/V,N,SUBCASE=1 $
PARAM //NOP/V,N,COUNT=1 $
PARAM //NOP/V,N,DES_VAR=1 $
$
LABEL LOOPVAR $
PARAM //ADD/V,N,OP2UN/V,N,ON1/V,N,OP2UN $
INPUTT2 /KX1,,,,/0/V,N,OP2UN $
PARAM //ADD/V,N,OP2UN/V,N,ON1/V,N,OP2UN $
INPUTT2 /KX2,,,,/0/V,N,OP2UN $
PARAMR //DIV/V,N,FACT/0.5/V,Y,EPSIL = 0.01 $
PARAMR //COMPLEX//V,N,FACT/0./V,N,ALPH $
ADD KX1,KX2/DK1/(1.0,0.0)/(-1.0,0.0) $
ADD DK1,/DKV1/ALPH $
$
LABEL LOOPSUB $
PRTPARM //0/C,N,DES_VAR $
PRTPARM //0/C,N,SUBCASE $
MATMOD ULV,,,,,/U,/1/V,N,SUBCASE $
PARAM //ADD/V,N,SUBCASE/V,N,ON1/V,N,SUBCASE $
MPYAD DKV1,U,/Y/1///2 $
FBS LLL,,Y/UV/ $
SDR1 USET,,UV,,,,,,KFS,KSS,/UGVS,,/1/C,N,STATICS $
SDR2 CASEDR,,,,EQEXINS,,,,,,,UGVS,,/,,OUGV1,,,/C,N,STATICS/
S,N,NOSORT2/V,Y,NOCOMPS=+1 $
OFP OUGV1// $
PARAM //SUB/V,N,COUNT/V,N,COUNT/V,N,ON1 $
COND OUT,COUNT $
REPT LOOPSUB,5 $
$
LABEL OUT $
PARAM //ADD/V,N,DES_VAR/V,N,DES_VAR/V,N,ON1 $
SETVAL    / / V, N, SUBCASE/ 1 $
SETVAL    / / V, N, COUNT/ 1 $
REPT LOOPVAR,2 $
$
END $
CEND $
BEGIN BULK
PARAM,EPSIL,0.01
ENDDATA
```

Figure 3

MSC/NASTRAN IMPLEMENTATION OF EIGENVALUE SENSITIVITY ANALYSIS

(VERSION 65)

```
BEGIN $
DBFETCH /MXX,KXX,LAMA,PHIX,/V,Y,SOLID///DBSET2 $
LAMX, ,LAMA/LMAT/-1 $
MATMOD PHIX,,,,,/U,/1/V,Y,COL $
PARAML LMAT//DMI/1/V,Y,COL/V,N,EVL $
PARAM //NOP/V,N,ON1=1  $
PARAM //NOP/V,N,OP2UN=15 $
$
LABEL LOOPVAR $
PARAM //ADD/V,N,OP2UN/V,N,ON1/V,N,OP2UN $
INPUTT2 /MX1,KX1,,,/0/V,N,OP2UN $
PARAM //ADD/V,N,OP2UN/V,N,ON1/V,N,OP2UN $
INPUTT2 /MX2,KX2,,,/0/V,N,OP2UN $
PARAMR //DIV/V,N,FACT/0.5/V,Y,EPSIL = 0.01 $
PARAMR //COMPLEX//V,N,FACT/0./V,N,ALPH $
ADD KX1,KX2/DK1/(1.0,0.0)/(-1.0,0.0) $
ADD DK1,/DKV1/ALPH $
ADD MX1,MX2/DM1/(1.0,0.0)/(-1.0,0.0) $
ADD DM1,/DMV1/ALPH $
PARAMR //MPY/V,N,FAC/-1./V,N,EVL $
PARAMR //COMPLEX//V,N,FAC/0./V,N,LAMD $
ADD DKV1,DMV1/MID/(1.0,0.0)/LAMD $
SMPYAD U,MID,U,,,/TOT/3////2////6 $
PARAML TOT//DMI/1/1/V,N,DLAM $
PRTPARM //0/C,N,DLAM $
REPT LOOPVAR,6 $
$
END $
CEND $
BEGIN BULK
PARAM,COL,1
PARAM,EPSIL,0.01
ENDDATA
```

Figure 4

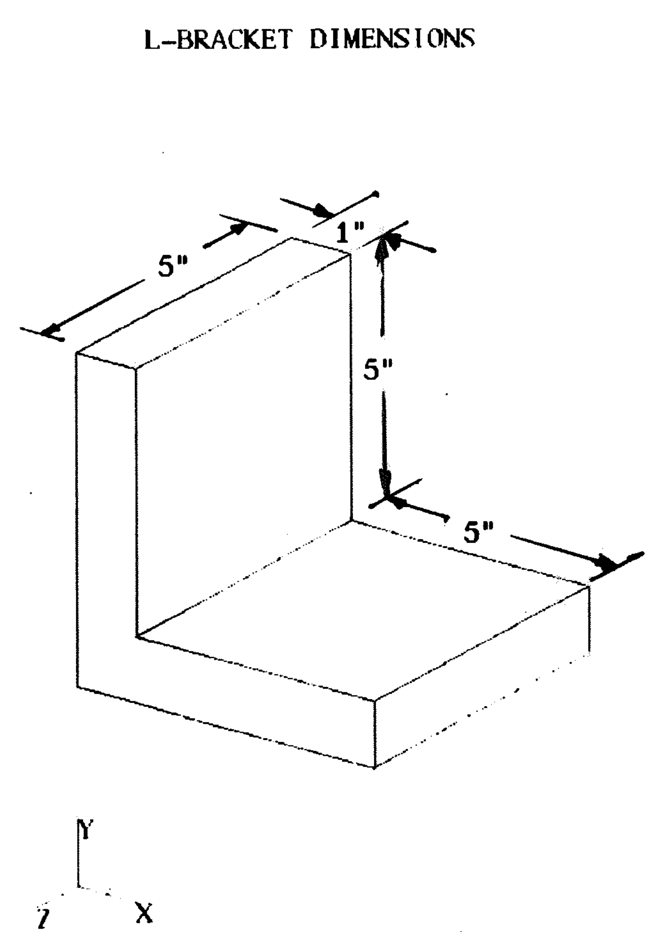


Figure 5

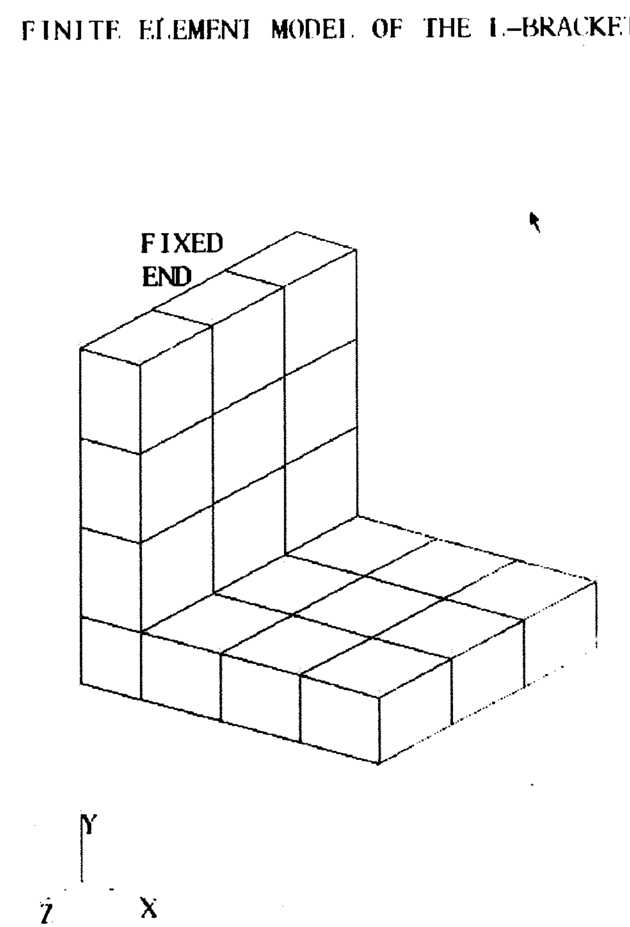


Figure 6

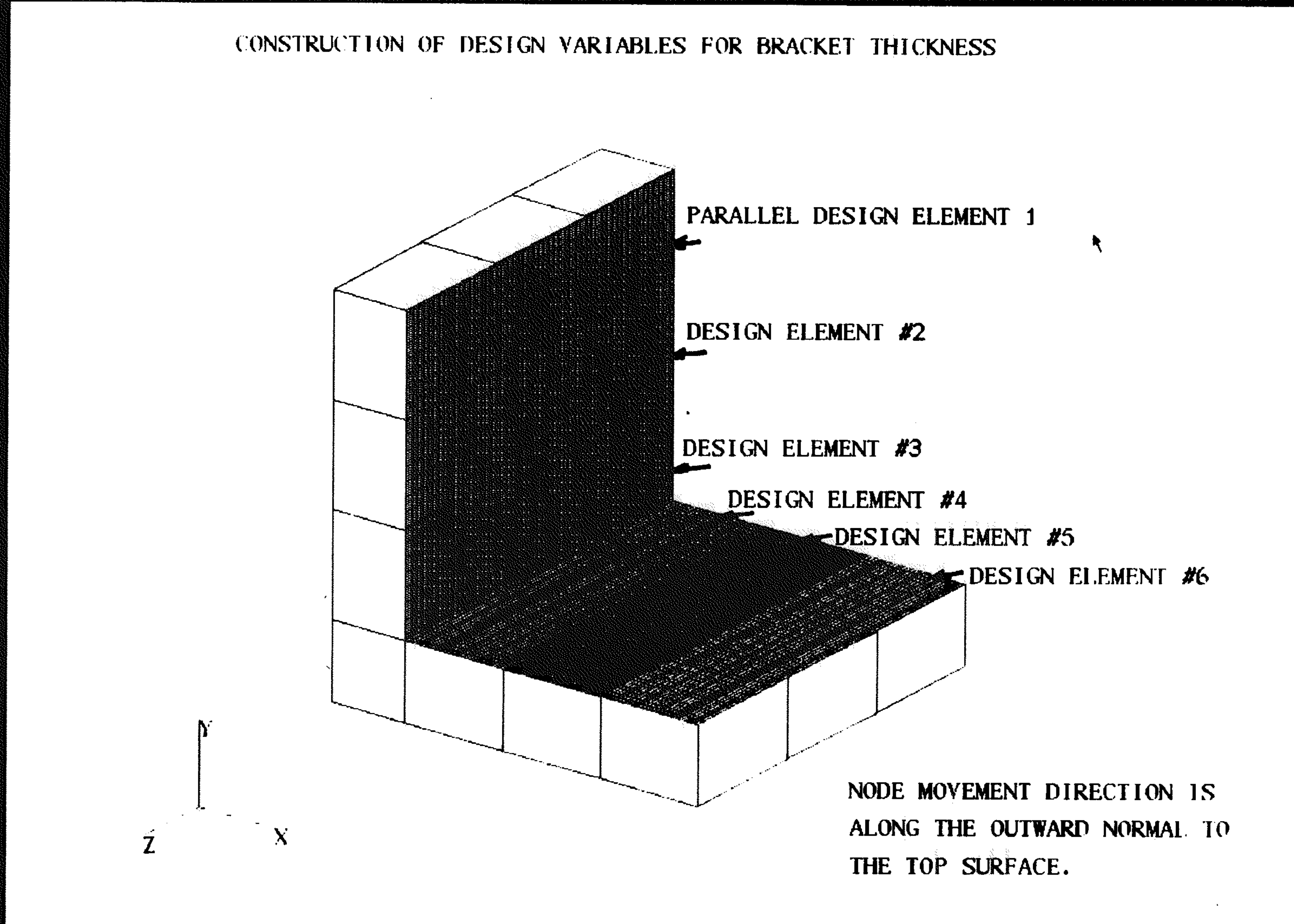
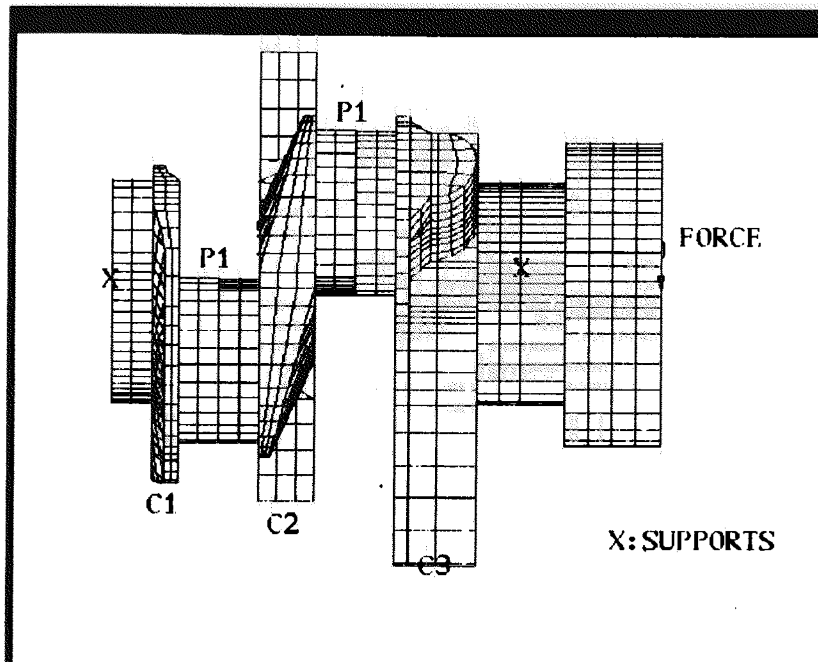

CONSTRUCTION OF DESIGN VARIABLES FOR BRACKET THICKNESS
PARALLEL DESIGN ELEMENT 1
DESIGN ELEMENT #2
DESIGN ELEMENT #3
DESIGN ELEMENT #4
DESIGN ELEMENT #5
DESIGN ELEMENT #6
Y
Z
X
NODE MOVEMENT DIRECTION IS
ALONG THE OUTWARD NORMAL TO
THE TOP SURFACE.

Figure 7

FINITE ELEMENT MODEL OF AN AUTOMOTIVE ENGINE CRANKSHAFT



$$\text{BENDING DEFLECTION SENSITIVITY} = \frac{\text{\% CHANGE IN LATERAL DEFLECTION}}{\text{\% CHANGE IN DESIGN VARIABLE}}$$

DESIGN VARIABLES: CHEEK THICKNESSES(C1,C2,C3) & PIN DIAMTER(P1)

Figure 8

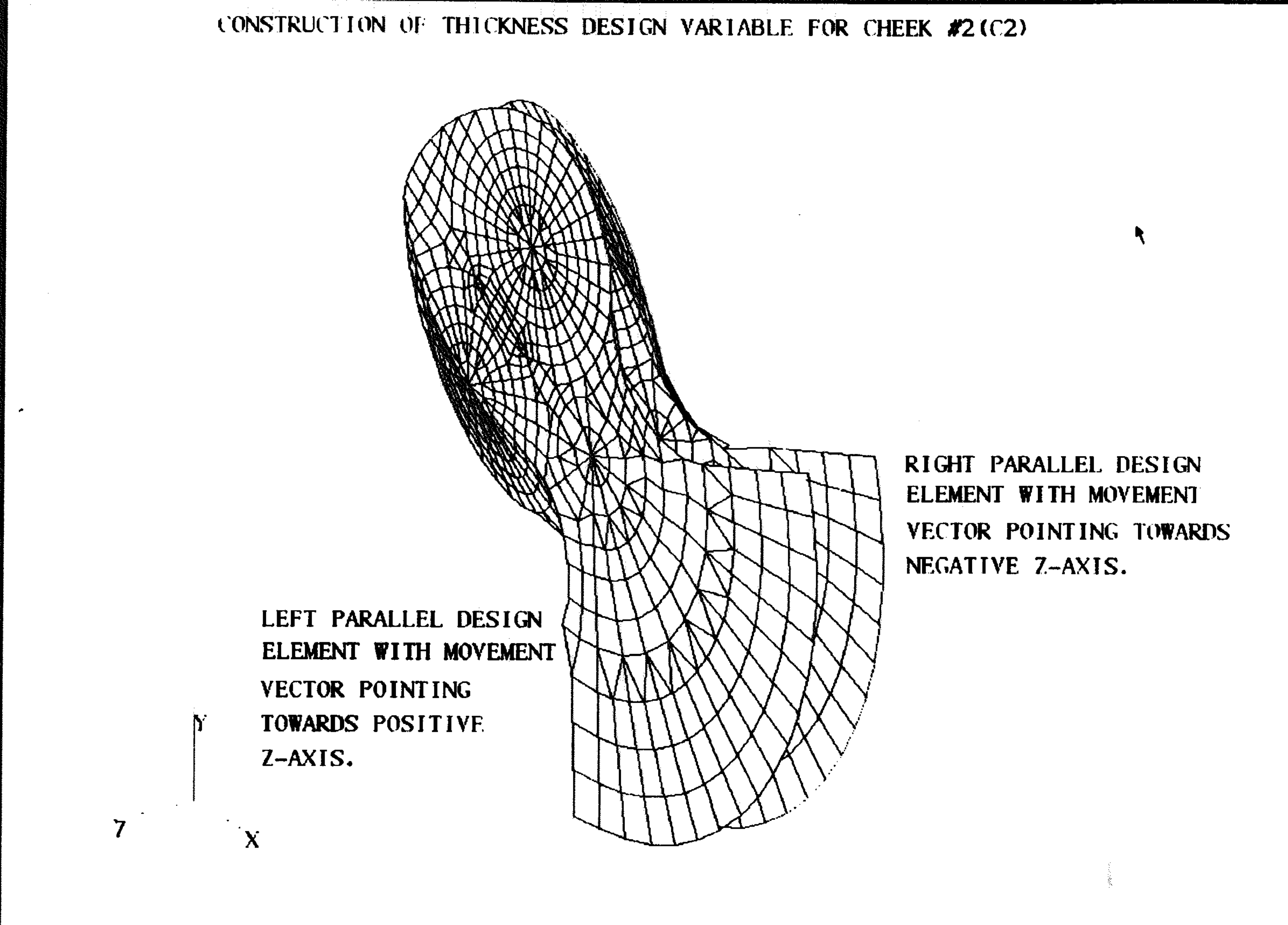
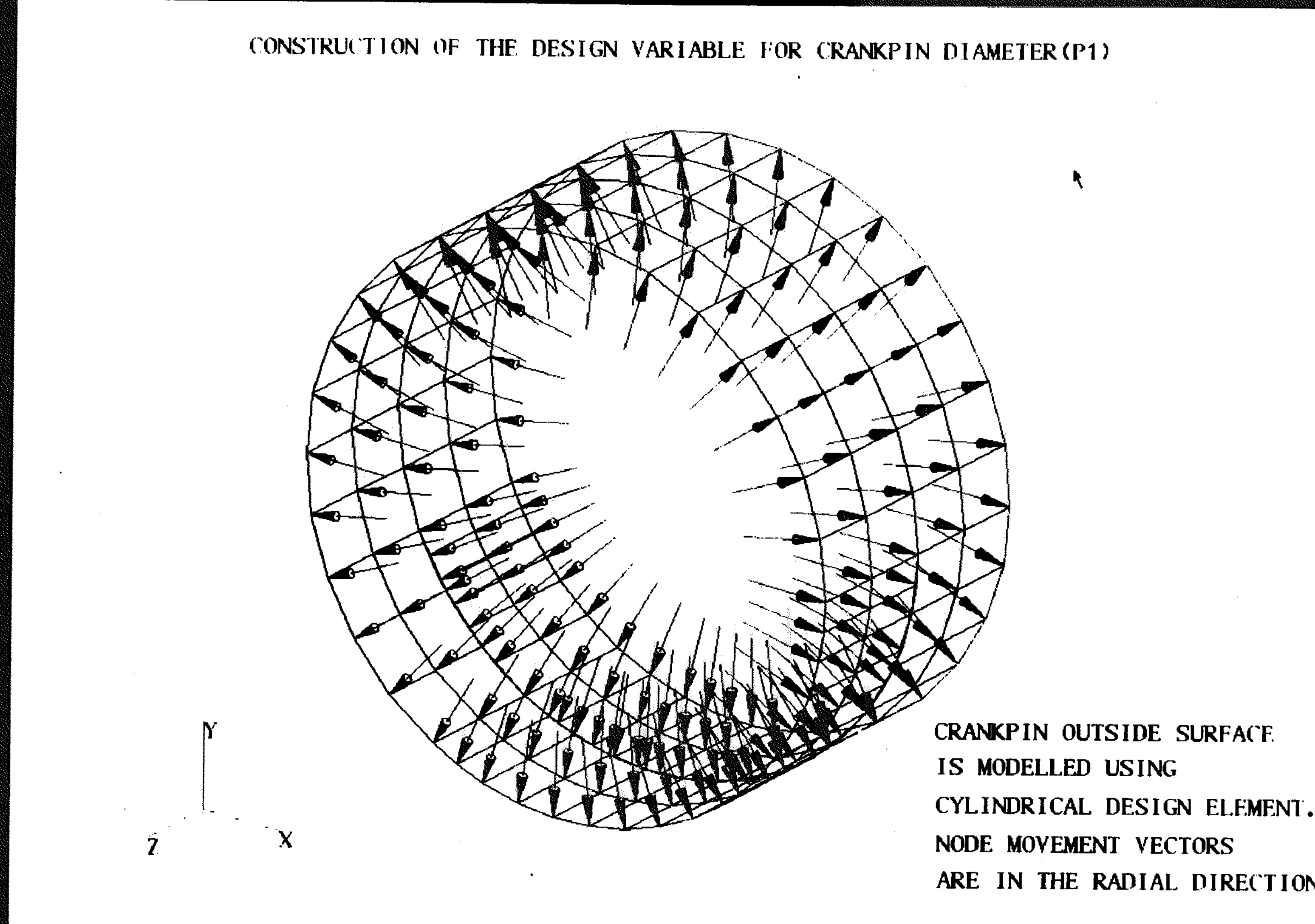

CONSTRUCTION OF THICKNESS DESIGN VARIABLE FOR CHEEK #2(C2)
RIGHT PARALLEL DESIGN ELEMENT WITH MOVEMENT VECTOR POINTING TOWARDS NEGATIVE Z-AXIS.
LEFT PARALLEL DESIGN ELEMENT WITH MOVEMENT VECTOR POINTING TOWARDS POSITIVE Z-AXIS.
Y
X

Figure 9



CONSTRUCTION OF THE DESIGN VARIABLE FOR CRANKPIN DIAMETER (P1)
Y
Z
X
CRANKPIN OUTSIDE SURFACE
IS MODELLED USING
CYLINDRICAL DESIGN ELEMENT.
NODE MOVEMENT VECTORS
ARE IN THE RADIAL DIRECTION

Figure 10