

CRANFIELD INSTITUTE OF TECHNOLOGY

School of Mechanical Engineering

Ph D Thesis

Academic Year 1978-81

K SHAM SUNDER

FINITE ELEMENT ANALYSIS OF CENTRIFUGAL IMPELLERS

Supervisor :

Dr R A Cookson
Head of Applied Mechanics Group

External Examiner :

Mr D A Thurgood
Chief Mechanical Engineer
British Aerospace Dynamics Group

October 1981

BEST COPY

AVAILABLE

Variable print quality

In the loving memory of my father
Sri Balishwaraiah Guptha and to
my beloved mother Smt Anasuyamma

SUMMARY

A three-dimensional method of stress analysis using finite element techniques is presented for determining the stress distribution in centrifugal impellers. It can treat all of the three types of loading possible in an impeller, viz centrifugal, thermal and fluid. The method has no known limitations with regards to the geometric factors such as asymmetry of disk, blade curvature, presence of a cover disk or shroud, single or double sided impeller etc.

A comparison of results with available experimental photoelastic results is presented with good agreement. The problem of the inter-blade bending effect, on the stress characteristics of an impeller, with relevance to the number of blades is studied in some depth. An insight into the effect of blade curvature on the stress characteristics of an impeller is also achieved.

As an extension of the above work, a method is proposed for the analysis of the dynamic behaviour of impellers, achieving a reasonable degree of success, particularly considering the limited period of time that was available for such an exercise.

ACKNOWLEDGEMENTS

I wish to express my heartfelt thanks to Dr R A Cookson for suggesting this interesting problem and supervising the project. This thesis would not have come to fruition without his constant encouragement and advice throughout the course of this work.

My sincere thanks to Mr J W Rait for his interest and critical appreciation throughout the duration of this project. He will undoubtedly find some of his suggestions directly implemented. Also my thanks to Dr R A Bannister and Dr S A Meguid for a number of useful discussions.

This work would not have been possible without the facilities and the efficient service made available by the Institute's computer Centre. Particular thanks to Mr Paul Verey for his cooperation and timely assistance.

It is with gratitude that I recall Mrs Terri Miflin, whose friendship and warmth is memorable. May I also recall my colleagues, Richard Stamp and others, for their contribution to a happy working environment. My thanks to Mr R W Brown for offering me timely help in drawing some of the figures and also to Mrs Amy Allen-Rowlandson for accepting the difficult task of typing the equations in this thesis.

I am grateful to the Commonwealth Commission in the United Kingdom for the financial support. My thanks to M/S Hindustan Aeronautics Ltd, India, for the study leave.

Finally, I wish to acknowledge my wife Shobha for her forbearance and help and the other members of our family in India for their encouragement, particularly after the unexpected loss of my father during my stay here.

CONTENTS

	Page No
SUMMARY	
ACKNOWLEDGEMENTS	
LIST OF FIGURES	
NOTATION	
CHAPTER 1 INTRODUCTION	1
1.1 General Description	2
1.2 Literature Survey	4
CHAPTER 2 FINITE ELEMENT FORMULATION	11
2.1 General Description	12
2.2 Choice Of Elements	12
2.3 Coordinate System	13
2.4 20-Node Isoparametric Brick Element	15
2.5 15-Node Isoparametric Prism Element	23
2.6 Load Vector Due To Rotation	24
2.7 Numerical Integration	28
2.8 The Principle Of Repeatability	33
2.9 Assembly And Solution	34
2.10 Evaluation Of Stresses	38
2.11 Thermal Loading	40
2.12 Fluid Pressure Loading	42
CHAPTER 3 PROGRAMMING CONSIDERATIONS	45
3.1 General Description	46
3.2 General Characteristics	46

3.3	Program Layout	48
3.4	Input Program	48
3.5	Frontal Solution	52
3.6	Output Processor	53
3.7	Extensions	54
CHAPTER 4	RESULTS	56
4.1	General Description	57
4.2	Accuracy And Convergence	58
4.3	Comparison With Photoelastic Results	59
4.4	Inter-Blade Bending	63
4.5	Impellers With Non-Radial Blades	65
4.6	Other Disk Configurations	66
4.7	Thermal Loading	67
4.8	Fluid Pressure Loading	68
CHAPTER 5	AN EXTENSION TO DYNAMIC ANALYSIS	70
5.1	General Description	71
5.2	Algorithms For Dynamic Analysis	71
5.3	Choice Of A Solution	73
5.4	Proposed Method	74
5.5	Computer Implementation	74
5.6	Results	78
CHAPTER 6	CONCLUSIONS AND RECOMMENDATIONS	81
6.1	Static Analysis	82
6.2	Dynamic Analysis	82
6.3	Further Research	83

REFERENCES	84
APPENDIX - A	90
FIGURES	96

LIST OF FIGURES

- 1.1 A typical centrifugal impeller
- 1.2 Disc terminology
- 2.1 Cylindrical system of coordinates
- 2.2 20-node isoparametric brick element
- 2.3 15-node isoparametric prism element
- 2.4 A sector of a ring (under centrifugal load) modelled with three 20-node elements
- 2.5 Repeatable structures
- 2.6 Optimal sampling points in a 20-node element
- 2.7 Optimal sampling points in a 15-node element
- 2.8 A brick element showing an infinitesimal surface area as a vector normal to the face
- 3.1 Overall flow diagram
- 3.2 Flow diagram of Input Program
- 3.3(a-g) Impeller Model-P Mesh-I
- 3.4 Flow diagram of Frontal Solution Program
- 3.5 Flow diagram of Output Processor Program
- 4.1 Hawker Siddeley Dynamics Limited impeller models
- 4.2(a-b) Impeller Model-P Mesh-I : stress and displacement characteristics
- 4.3(a-c) Impeller Model-P Mesh-II
- 4.4(a-b) Impeller Model-P Mesh-II : stress and displacement characteristics
- 4.5(a-c) Impeller Model-P Mesh-III
- 4.6(a-b) Impeller Model-P Mesh-III : stress and displacement characteristics
- 4.7(a-c) Impeller Model-P Mesh-III : stress characteristics (Epoxy as material)

- 4.8 (a-d) Impeller Model-N Mesh-III
- 4.9 (a-d) Impeller Model-N Mesh-III : stress and displacement characteristics (Epoxy as material)
- 4.10 (a-c) Impeller Model-R Mesh-III
- 4.11 (a-c) Impeller Model-R mesh-III : stress characteristics (Epoxy as material)
- 4.12 Disk from Impeller Model-P Mesh-III
- 4.13 (a-c) Disk from Impeller Model-P Mesh-III : stress and displacement characteristics
- 4.14 Disk from Impeller Model-N Mesh-III
- 4.15 (a-c) Disk from Impeller Model-N Mesh-III : stress and displacement characteristics
- 4.16 (a-b) Impeller Model-P Mesh-III : models with variation in number of blades
- 4.17 (a-h) Impeller Model-P Mesh-III : stress and displacement characteristics with variation in number of blades
- 4.18 Inter-blade bending effect in Impeller Model-P
- 4.19 (a-b) Impeller Model-P Mesh-III : models with variation in blade curvature and number of blades
- 4.20 (a-h) Impeller Model-P Mesh-III : stress and displacement characteristics with variation in blade curvature and number of blades
- 4.21 (a-c) Impeller Model-P with cover disk Mesh-III
- 4.22 (a-b) Impeller Model-P with cover disk Mesh-III : stress and displacement characteristics
- 4.23 (a-b) Impeller Model-R Mesh-III with 7 blades representing a symmetric half of a double sided impeller : stress and displacement characteristics
- 4.24 (a-d) Impeller Model-P Mesh-III with temperature gradient : stress and displacement characteristics
- 4.25 (a-b) Armco test impeller and fluid pressure distribution
- 4.26 (a-d) Armco test impeller : finite element mesh
- 4.27 (a-e) Armco test impeller : stress and displacement characteristics for centrifugal and fluid pressure loadings

- 5.1 Overall flow diagram for dynamic analysis
- 5.2 A repeatable sector of a ring idealised with two brick elements for dynamic analysis
- 5.3(a-m) Impeller Model-P Mesh-I 8 blades : vibratory mode shapes
- A.1 General form of stiffness and mass matrices of a typical structure with four repeated substructures
- A.2 Form of the decoupled stiffness and mass matrix after a Fourier transformaton
- A.3 Cyclically repetitive structure
- A.4 A typical repetitive sub-structure

NOTATION

$[B]$	Strain matrix
$\{b\}$	Body force vector
$[D]$	Elasticity matrix
$D1, D2, D3$	Constants (in elasticity matrix)
E	Young's modulus
$\{F\}$	Vector of forces
$\{f\}$	Element load vector
$f(\xi)$	A general function of
H_i	Weight coefficient
$[J]$	Jacobian transformation matrix
$[K]$	Stiffness matrix
K_{ij}	A sub-matrix of stiffness matrix
$[L]$	Differential operator matrix
$L1, L2, L3$	Area Coordinates
$[M]$	Mass matrix
$[N]$	Matrix of shape functions
N_i	A shape function
n	Number of integration points
$[P], [P']$	Matrices used in stress smoothing equations
p	Surface fluid pressure
$[Q]$	Stress matrix
$\{R\}$	Vector of reactions
$[S]$	Stress smoothing matrix
T	Temperature
(r, z, θ)	Cylindrical coordinates

u, v, w	Displacements (radial, axial, tangential)
dA	Derivative of surface area
dV	Derivative of volume
ν	Poisson's ratio
$\{\delta\}$	Vector of displacements
$\{\sigma\}$	Vector of stresses
ϵ_0	Initial strain
α	Coefficient of thermal expansion
ρ	Mass density of material
ω	Angular speed of rotation (radians/sec)
$[\phi]$	Eigenvector matrix
ξ, η, ζ	Natural coordinates
$\phi(\xi, \eta, \zeta)$	A general function
$\epsilon_r, \epsilon_z, \epsilon_\theta$	Normal strains
$\nu_{rz}, \nu_{r\theta}, \nu_{z\theta}$	Shear strains
$\sigma_r, \sigma_z, \sigma_\theta$	Normal stresses
$\tau_{rz}, \tau_{r\theta}, \tau_{z\theta}$	Shear stresses
∂	Partial derivative operator
$\rightarrow$	Indicates a vector

CHAPTER 1

INTRODUCTION

1.1 GENERAL DESCRIPTION

An impeller with lateral vanes (Figs 1.1 and 1.2) presents many difficulties for a rigorous analysis of stress and strain. The limitations imposed by stress considerations are often the governing factor in the optimum design of centrifugal impellers. Application of a factor of safety, in order to provide insurance for empirical methods, often results in the use of too much material or too many stages, or more usually both. The types of failure that can occur in an impeller besides a complete rupture, include certain elastic and inelastic deformations causing unsatisfactory operation. For example, excessive growth of an impeller bore may allow the impeller to work loose from the shaft causing rotor imbalance. Elastic or plastic deformations of the cover at the eye can cause a rub on the air seal, and excessive growth of the impeller can result in fouling of the casing. Whenever plastic deformation takes place, the load is shifted to other parts which may not have been designed to take this increased loading.

Owing to these factors and the extremely high speeds encountered in current day impellers (of the order of 30,000rpm), a rigorous analysis for stress and displacement is most desirable. The first step in such an analysis is the determination of stresses and displacements induced by the centrifugal forces alone. The analysis can then be extended by the consideration of stresses due to thermal conduction and fluid pressure distribution.

In the past, a number of methods have been presented for the stress analysis of impellers. They range from the early classical solutions for the symmetric disk, to the recent finite difference and finite element analyses for impellers with radial blades. In the early methods the presence of blades are accounted for by a simple assumption of distributing the mass of the blades uniformly over the disk, either by increased specific gravity or thickness. The later concepts include achieving elastic balance between disk, blades, and sometimes cover plates; or replacing the blades with a fictitious orthotropic disk having equivalent radial strength but no hoop strength.

In all such cases, the basic assumption is, that the stress and displacement characteristics in the disk are independent of the angular coordinate, with the apparently justifiable reason, that when the number of blades is large, the periodic component due to the blades can be neglected. This simple assumption reduces the basic three-dimensional problem to a two-dimensional axisymmetric problem facilitating the use of many published procedures. Whilst the above assumption makes it clear that impellers with few blades are not amenable to such methods, even for impellers with a relatively large number of blades, great care is required while using these solutions, since other factors such as the shape of the disk and the size of the blades influence the strength of the periodic component.

Recently, several assumptions employed in two-dimensional solutions, such as the Kirchhoff-Love hypothesis, neglect of axial and shear stresses have been shown erroneous, casting doubts on the accuracy of the methods. The literature points to the use of a three-dimensional finite element technique as a means for the realisation of a complete and satisfactory solution, although raising fears with regard to the large input data involved and the need for larger computer resources.

In this thesis an effort is made to adopt a three-dimensional finite element method and present a computer package which minimises the input data and computer resources required, suiting better the needs of present day industry. In addition to the traditional radial impeller, which was the model investigated by most previous workers, the present work covers many other important cases, for instance the presence of non-radial blades, cover plates, and shrouds. The analysis has no known limitations with regard to the number of blades or asymmetry of the disk. The heat conduction and fluid pressure loading cases have been solved as extensions of the above problem.

In the available time, attempts have been made to extend the work to cover the dynamic behaviour of impellers, achieving a reasonable degree of success.

1.2 LITERATURE SURVEY

The stress analysis of an axisymmetrical body under the influence of centrifugal body forces has been of considerable importance to the industry for a very long time. One can find literature on the subject dating back to as early as the year 1850, when Maxwell (Ref 1) considered the classical problem of the rotating disk of uniform thickness. It will obviously be impracticable to review all the conceivable methods presented since then. References 2 to 8 are some of the early publications on the subject. Here an attempt is made to review the chronological development of the methods presented on the subject, with the help of some representative sample, while concentrating on some useful developments.

Jaburek (Ref 9), in the year 1953, presented a solution for radially vaned impellers, by considering the vaned disk as approximated by an orthogonally anisotropic circular plate. Stresses due to centrifugal force were presented in the form of superposition of virtually plane stresses and super imposed bending stresses. He presented results for an impeller, but with no verification. Glessner (Ref 10), in the following year, presented a method characterised by the assumption of an elastic balance between the blades, disk, and cover of an impeller under rotation. It is not an exact method and uses certain approximations. The results obtained using the above method were verified with the aid of an experimental investigation of the bursting speeds for a number of impellers. The analytical technique was found to predict the burst speed with reasonable accuracy. However, as became known later, the prediction of bursting speed based on maximum stress is unreliable. Kobayashi and Trumpler (Ref 13), presented a three-dimensional treatment for the calculation of elastic stresses in a rotating disk of general profile. The method consisted of a linear combination of the states of stress of a basic plane-strain problem of a long rotating cylinder, and two residual problems. The residual problems were, one the elimination of surface tractions acting on a disk which is cut from the rotating cylinder, and the other the imposition of prescribed displacements along the disk bore. A finite difference form was used for the solution. Though the results presented were for disks without blades, the effects of asymmetry of the disk on the stresses and the existence of axial stresses at the bore in disks with a hub were clearly demonstrated. These two factors later attracted considerable attention from the researchers in the field. The different stresses on the front and rear of the impeller were presented separately, showing the effect of disk bending.

Schilhansl (Ref 14), showed the errors involved in distributing the mass of the blades on the disk of a radial flow impeller, by considering the differential growth between the blades and the disk for a simple case. He presented a two-dimensional finite-difference method of analysis for single sided radial flow impellers. Compatibility of the displacements and rotations at the blade-disk junction, was achieved to establish the bending stresses, and hence the evaluation of total stress

distribution on the front and rear disk surfaces and the blade leading edges. Though Schilhansl did not pay any special attention to the problem of asymmetry of the disk, the theory deals with disk asymmetry. As became known later, the Schilhansl technique is one of the most accurate of the two-dimensional methods published to date. The main assumptions used by Schilhansl were

1. Stress and displacement characteristics are independent of angular coordinate for impellers with a relatively high number of blades (12 or more).
2. Radial displacement varies linearly along the axial coordinate (Kirchoff-Love hypothesis).
3. Shear and axial stresses are considered zero. ??

Results for a typical radial impeller were presented. The radial stresses at the disk-blade junction were seen to be discontinuous, while the displacements were continuous. The following year, Swansson (Ref 15) adopted the above analysis and replaced the finite difference equations with basic differential equations to attain better accuracy. The fourth-order Runge-Kutta method was used for integration of the above equations. The predicted bursting speeds of impellers, based on the maximum stress obtained from the method, were 10 to 15% lower than the burst test results. Swansson explained the discrepancy by stating that the burst test results were not consistent; and listed a number of factors such as the stress pattern throughout the disk, the state of stress, the stress gradients, ductility and strain-hardening which influence the bursting speed making its prediction based on maximum stress unreliable. Swansson obtained very low stresses for the blade, and he suggested that for blade stress at the root a stress concentration factor would be required. This is a suggestion originally made by Glessner (Ref 10) with a stress concentration factor of 1.35 obtained from tests on cast aluminum alloy of low ductility. The occurrence of compressive stresses close to the centre on the front face of the disks, as calculated by his method, are said to be due to the assumption that straight lines parallel to the axis remain straight after deformation.

Rimrott and Bell (Ref 16) presented a theory for radial impellers, also in 1963 by assuming a parabolic variation of radial displacement along the axial coordinate in place of the linear variation suggested by Schilhansl. His other assumptions were that circular sections become paraboloid upon stressing, and that radial strain is continuous but that the circumferential strain at the blade-disk junction is not continuous. The technique proposed by Rimrott and Bell senses the presence of asymmetrical protrusions and takes their influence on bending and radial growth into consideration.

Also during 1963 an investigation report (Ref 17) on the stress distribution of impeller wheels was published by Wright-Patterson Air-Force Base. In that, a theory was presented based on rotating

shells, in which the differential equations for smooth shells were altered to include, blades free of tangential stress, the influence of cantilever rings representing the hub, and any additional loads. A ring is considered free of radial stress, which is probably applicable for the case of thin rings, and is replaced by an effective length of virtual equivalent ring with constant tangential stress in the axial direction. Finally, the differential equations were transformed into equations of differences and were solved by use of matrix calculus. A study of various parameters such as the slope of the disk (angular inclination of disk diaphragm) was made. The results given for variation in the number of blades showed almost no influence of inter-blade bending. The report also contains some information regarding a supporting experimental investigation. Experimentally, strain gauging on the surface of the disk and the application of the photoelastic method in conjunction with a stroboscope for blades was used. The report stated that the photoelastic results were qualitative, and that the strain gauge readings were unreliable as proper temperature compensation was not achieved. However, the gravest error in the reported experiments would appear to lie in the assumption that a number of blades (15 to 18) could be represented by four equivalent thick blades. This particular assumption would result in large distortion of the disk stresses, as four blades would result in a very high order of inter-blade bending stresses, probably invalidating the whole exercise.

A major investigation in this area, was carried out by Thurgood and Givan at Hawker Siddeley Dynamics Limited. In their first report (Ref 18), published in January 1967, they presented a finite-difference theory for the stress analysis of asymmetric profile rotating disks, including the effects of blade loading and stiffening in the case of single sided radial impeller. The stress results obtained were compared with experimental results, using the photoelastic stress freezing technique. Based upon a previous experience (Ref 19), where out-of-plane deflections of significant order were found to exist for cases of relatively thin disks with a high degree of asymmetry; they presented an iterative procedure wherein the out-of-plane deflections are added to the asymmetry of the disk, effectively changing it for the next iteration, and the iterations are continued until the input and output deflections agree within some accuracy. However, they suggested that when blade stiffening is included and the blades are deep, the out-of-plane deflections are small and may be neglected thus avoiding the iterative procedure. They added that this iterative procedure should be included for all unstiffened disks except those with a low degree of asymmetry. For the first time an extension was attempted to include the effect of inter-blade bending, highlighted by Williams and Harries (Ref 20) and their photoelastic investigation of a seven bladed model fan, which indicated inter-blade bending stresses of magnitude comparable with the disk bending stresses due to asymmetry. The extension was achieved by computing the difference in strain energies between a bladed stiffened disk and an unstiffened disk, and then equating it to that for an un-bladed disk displaced sinusoidally between blade root locations. The final stress distributions were obtained by superimposing the sinusoidal distributions and the stiffened disk distributions. However, this extension was not found to give

satisfactory results and hence was not used in their program for the calculation of stresses.

At this point, it may be worth describing the concept of inter-blade bending. In the case of impellers with blades on one side, the circumference of the disk deflects axially in the form of half sine wave between blade root locations. This effect is at its greatest at the outer radius and disappears towards the inner radius. It is quite pronounced for impellers with relatively few blades, and results in bending stresses of considerable magnitude.

Thurgood and Givan reported that inter-blade bending is not negligible even for impellers with a relatively large number of blades, (say seventeen). They based this conclusion on their experimental results. The effect of inter-blade bending is more pronounced on tangential stresses than on radial stresses. They considered the presented theoretical analysis to be inadequate for disks with relatively large blades, unless a large portion of the blade is assumed to be non-load carrying. They stated that the stress distributions were extremely susceptible to changes in profile both of disk and blades; and cast considerable doubt on the distributions obtained by Rimrott and Bell (Ref 16), where rapid changes of stress were demonstrated, with peaks almost amounting to discontinuities, based on a very limited number of radial stations. The frozen stress results they reported were considered satisfactory, but they added that there was room for improvement. The theory presented included stresses due to temperature gradient but these were not calculated.

Thurgood (Ref 21), in discussing the problems associated with photoelastic model testing, highlighted the effect of material properties on the stress characteristics of rotor and showed that the Poisson's Ratio has significant influence, while the effect of Young's Modulus was less. If the stresses in a steel prototype were assumed to be directly proportional to the stresses in the epoxy model, it would result in slightly over-estimating the stresses at the bore, while grossly under-estimating the stresses in the outer radius. For the case of impellers with a few blades, where inter-blade bending would result in high circumferential stresses, it is necessary to apply a correction factor, which is a function of radius to the model stresses.

The second report on asymmetric impellers by Thurgood and Givan (Ref 22), was based on a recommendation by Smith (Ref 23). They evaluated the four analytical techniques proposed by references 14,15,16, and 18. This evaluation was carried out by calculating the stress distributions produced within eighteen different rotors by rotation. For thirteen of the eighteen cases the calculated values were evaluated against photoelastic stress-frozen test results. The range of rotors examined extended from a symmetrical unvaned disk to a highly asymmetrical vaned impeller. The stress distributions were presented in a systematic and non-dimensional format, which has been adopted for the

presentation of the results of the present investigation. The rotors examined in reference 22 were bored and assumed to be made from Epoxy resin; and the vanes of the vaned rotors were assumed radial, trapezoidal in cross section, unshrouded and on one side of the disk only. While all the four methods gave satisfactory results for un-vaned symmetrically disposed disk profiles, their own method (Ref 18) was found to give substantially higher stress levels in the case of un-vaned asymmetrically disposed disks. The method of reference 18 is applicable only for flat back disks or for disks with minor deviations from that condition. For cases where a high degree of coning was involved, only the methods of references 14 and 15 were satisfactory. For the case of vaned asymmetrically disposed disk profiles all the methods predicted similar results, except in models with a high degree of coning where again their own method resulted in much higher stresses. While the results were more representative for impellers with a large number of blades, they were hardly satisfactory for impellers with relatively few blades. Even for the case of impellers with a large number of blades, the calculated results for the front surface were not in agreement with the experimental ones. Obviously, this is to be expected, since for an impeller with a large number of blades, the influence of the blades on the stress characteristics of the disk, is considerable on the front surface, and evens out over the angular coordinate as it reaches the rear surface of the impeller; making it possible for a two-dimensional analysis to approximate more closely to the correct values on the rear face.

Chan and Henrywood (Ref 24), pointed out that the evaluation carried in the above report is so inconclusive, as to leave doubts on the reliability of the two-dimensional methods. They presented a three-dimensional finite element solution, for radial impellers, using a specially developed sector element to represent the disk. The sector element is triangular in cross-section and uses a Fourier formulation enabling the analysis of only one segment of the impeller. The blades were assumed flat and to lie in radial planes and are represented by triangular membrane elements. Scope of the analysis is limited to radial impellers under rotation, and cannot be extended to consider fluid loading as the blades are represented by membrane elements. Results obtained were compared against photoelastic results and Schilhansl's finite difference results given in reference 22. Stresses presented for a seven bladed model were better than the finite-difference ones, proving the capability of a three-dimensional finite element solution to take inter-blade bending into account. However, the stresses presented for a seventeen bladed model were poorer, raising doubts on the accuracy achieved by their finite element model. Probably, one could attribute the apparent errors to the accuracy achievable by the early generation finite elements and concepts in use at that time. The results presented (Ref 24) were for Poisson's Ratio of 0.34, as their solution cannot accept the higher ratios (nearing 0.5), appropriate to the photoelastic models. With straight edged triangles, the curved surfaces are difficult to model and hence a finer mesh is required. The later generation isoparametric elements allow much better modelling, while achieving improved accuracies with relatively coarser mesh. The Fourier approximation used (Ref 24) can be

replaced with the much simpler Repeatability concept, while removing the restriction of the analysis to radial bladed impellers only. The banded form of solution used at that time can be replaced with the popular and more efficient Frontal solution achieving optimal use of computer resources. These and other relevant factors will be discussed in the later sections of the thesis.

Patton (Ref 25), presented an experimental programme of stress analysis using the brittle-lacquer and strain-gauge techniques applied to a range of centrifugal fan impellers. He concluded that the local and bending stresses are of the same order as the direct stresses predicted by conventional methods, and proposed a more sophisticated theoretical analysis, with one possible approach being a finite element solution.

Recent contribution to the literature on the subject, is that of Kikuchi (Ref 26). Following a proposal given in two previous Russian publications (Refs 27 and 28), he presented a method in which the radial blades were replaced by a fictitious solid disk, made of orthotropic materials, which give a radial strength equivalent to that of replaced blades, but possessing no hoop strength. With this modifying assumption, the whole impeller is treated as the simple case of an axisymmetric disk under rotation, using a conventional finite element method. He also presented photoelastic stress-frozen results for three simple impellers, with identical disks but having a different number of blades. The disks were of flat back shape and the blades were of constant thickness.

Kikuchi's significant contribution comes as a thorough examination of the various assumptions employed by previous workers in their two-dimensional formulations. He proved that the Kirchhoff-Love hypothesis, 'linear variation of radial displacement along axial coordinate', is not valid for rotors with large bore thickness. The actual stress distribution is non-linear and differs more where there is a rapid change in axial thickness. However, the parabolic variation assumed by Rimrott and Bell (Ref 16) is found to be even more subject to error, and the linear variation is still considered to be a better choice for two-dimensional finite-difference formulations. He stated that the linear variation becomes increasingly inadequate as the number of blades increases, and is undesirable for double-sided impellers where no out-of-plane bending exists. For rotors with a conical profile, the variation of radial displacement is linear but is still not the same as the one assumed by Schilhansl (Ref 14). Kikuchi pointed out that Rimrott and Bell were wrong in assuming that their propositions would result in zero shear stress. He confirmed the presence of axial stress, in rotors with large bore, as predicted by Kobayashi and Trumpler (Ref 13). By considering the case of disks with multiple point shear and radial loading, he concluded that the non-axisymmetric stress distributions are not only influenced by the number of loads (or blades), but also the magnitude of loads and the geometry of the disk. This raises more doubts on the assumption that the stress distribution could be considered independent of angular coordinate for 12 or more blades.

In Kikuchi's analysis, though the disk replacing the blades is assumed free of hoop strength, some tangential stresses are produced but are disregarded. Like all other two-dimensional methods, the analysis produces a discontinuity in the stresses at the blade-disk junction. His statement that the maximum circumferential stress at the bore on the rear side of an impeller decreases with the increase in the number of blades, would at first sight appear to be questionable. For a typical impeller with variable thickness blades, the orthotropic properties of the fictitious disk replacing the blades will change not only in the radial direction but also in the axial direction. That variation could result in the calculation of equivalent orthotropic properties at too many points, resulting in too many fictitious materials for finite element analysis. It would appear to be difficult to adopt the analysis (Ref 26) to deal with fluid loading since the blades are replaced by a fictitious disk. Although the technique employed finite element analysis, it has shown only a marginal improvement over the finite difference technique of Schilhansl. Kikuchi recommended the use of a three-dimensional finite element technique for greater accuracy in analysing the stresses produced in present day impellers, and hence increased confidence in meeting the greater demands placed upon current components. He suggested that the aims of such an analysis should be, to minimise the requirements of the computer core and time, and the human effort for input-data preparation.

The above literature survey leads one to the conclusion that the whole series of methods presented so far have not been adequate for a satisfactory analysis of impellers with a varied number of blades and disk configurations to be made. Following Kikuchi's recommendations, an attempt has been made to develop a satisfactory three-dimensional finite element solution and a computer package, which is hoped to suit better the needs of present day industry.

CHAPTER 2

FINITE ELEMENT FORMULATION

2.1 GENERAL DESCRIPTION

The presence of lateral vanes in an impeller necessitates a three-dimensional solution for stress analysis. Usually, the displacement and stress pattern due to centrifugal, thermal, and fluid loading is repetitive over all the blade segments. It is ideal and sufficient if only one blade segment is analysed with suitable equivalent boundary conditions. The simplest and most elegant way of achieving this in any finite element formulation, for static stress analysis, is to use the principle of repeatability as illustrated in references 29 and 30. This principle proposes assigning the same destinations, during structural assembly, for the corresponding nodes on the repeating boundaries of the individual segments of an impeller considered for analysis. To apply the principle, it is essential and also convenient to treat the problem as a cylindrical coordinate system i.e., R-Z- θ coordinates.

In the finite element formulation adopted, a repeatable segment of an impeller is considered, in cylindrical coordinates. The well known finite element displacement formulation (Ref 30) is adopted. A frontal solution (Ref 31) is used for the assembly and reduction of the resulting large set of (load-displacement) simultaneous equations. The novel concepts of 'minimum integration' and 'optimal sampling of stresses' are used to improve the quality of the results obtained. Throughout the formulation, the well established relations and notations of reference 30 have been used.

2.2 CHOICE OF ELEMENTS

One of the main factors affecting the accuracy and quality of the calculated results, for any finite element solution, is the suitability of the elements chosen to represent the physical configuration and the actual displacement and stress distributions within the component under analysis. Often the choice made is at best an engineering judgement based on the information available on the elements in the literature, and a priori qualitative assessment of the stress distribution expected in the problem considered. The following considerations were made while choosing the elements for the current formulation.

A typical impeller in general has a solid core at the hub and a plate or shell type structure elsewhere in the disc and the blades. The stresses vary non-linearly in the hub region and are of plane stress type with superimposed bending stresses in the disc and the blades. The bending stresses arise due to the asymmetry of the disc. The ideal case would be to model the hub with solid isoparametric elements (Refs 32, 33, 30) and the disc and the blades with super-parametric thick shell elements (Refs 34, 35). Unfortunately, the nodal degrees of freedom for the two types of elements are incompatible and there would be considerable difficulty experienced in connecting them. One means of overcoming this incompatibility can be the employment of a third set of transition elements (Ref 36) to join the two

parent type elements. This method will inevitably increase the complexity of the formulation and the computer implementation. One alternative is to use only one type of element for the complete impeller. As the shell elements would be unsuitable for the hub region, it was decided to model the whole impeller with solid elements. This choice obviously increases the size of the problem to a certain extent, since more degrees of freedom are involved with solid elements. However, the quality of the final results are not expected to suffer as the solid isoparametric elements have been proved to give good results, even when one of the dimensions is considerably smaller than the others (Ref 32). Isoparametric elements can also cater for a relatively high order of bending stress, particularly when the concepts of 'minimum integration' and 'optimal sampling' are adopted.

In view of the above reasoning, a 20-node isoparametric quadratic brick element is used as a parent element, and the 15-node isoparametric quadratic prism element is provided as a filler element in modelling.

2.3. COORDINATE SYSTEM

The cylindrical coordinate system, along with the displacements considered at a point, are shown in Fig 2.1.

The three-dimensional strain-displacement relationship in a cylindrical system of coordinates is given by (Ref 30),

$$\begin{Bmatrix} \epsilon_r \\ \epsilon_z \\ \epsilon_\theta \\ v_{rz} \\ v_{r\theta} \\ v_{z\theta} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial u}{\partial r} \\ \frac{\partial v}{\partial z} \\ \frac{u}{r} + \frac{1}{r} \frac{\partial w}{\partial \theta} \\ \frac{\partial u}{\partial z} + \frac{\partial v}{\partial r} \\ \frac{1}{r} \frac{\partial u}{\partial \theta} + \frac{\partial w}{\partial r} - \frac{w}{r} \\ \frac{1}{r} \frac{\partial v}{\partial \theta} + \frac{\partial w}{\partial z} \end{Bmatrix} \quad \dots \quad (2.1)$$

where ϵ_r , ϵ_z and ϵ_θ are the normal strains,

v_{rz} , $v_{r\theta}$ and $v_{z\theta}$ are the shear strains, and

u , v and w are the displacements at a point.

The stress-strain relationship for the case of an isotropic material is given by,

$$\begin{Bmatrix} \sigma_r \\ \sigma_z \\ \sigma_\theta \\ \tau_{rz} \\ \tau_{r\theta} \\ \tau_{z\theta} \end{Bmatrix} = [D] \begin{Bmatrix} \epsilon_r \\ \epsilon_z \\ \epsilon_\theta \\ \nu_{rz} \\ \nu_{r\theta} \\ \nu_{z\theta} \end{Bmatrix} \quad \dots \quad (2.2)$$

where $[D]$, the elasticity matrix is given by,

$$[D] = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1 & \frac{\nu}{1-\nu} & \frac{\nu}{1-\nu} & 0 & 0 & 0 \\ & 1 & \frac{\nu}{1-\nu} & 0 & 0 & 0 \\ & & 1 & 0 & 0 & 0 \\ & & & \frac{1-2\nu}{2(1-\nu)} & 0 & 0 \\ \text{SYM} & & & & \frac{1-2\nu}{2(1-\nu)} & 0 \\ & & & & & \frac{1-2\nu}{2(1-\nu)} \end{bmatrix} \quad \dots \quad (2.3)$$

In the above,

σ_r , σ_z and σ_θ are the normal stresses

τ_{rz} , $\tau_{r\theta}$ and $\tau_{z\theta}$ are the shear stresses

E and ν are the material's Young's Modulus
and Poisson's Ratio respectively.

2.4 20-NODE ISOPARAMETRIC BRICK ELEMENT

The twenty node brick element in the local natural coordinate system and a representative mapped element in the actual coordinate system are shown in Fig 2.2.

The shape functions for the element (Ref 30) are given as follows :

Corner Nodes

$$N_i = \frac{1}{8} (1 + \xi_0) (1 + \eta_0) (1 + \zeta_0) (\xi_0 + \eta_0 + \zeta_0 - 2)$$

Mid-Side Nodes

a) for $\xi_i = 0$ $\eta_i = \pm 1$ and $\zeta_i = \pm 1$

$$N_i = \frac{1}{4} (1 - \xi^2) (1 + \eta_0) (1 + \zeta_0)$$

b) for $\eta_i = 0$ $\xi_i = \pm 1$ and $\zeta_i = \pm 1$

$$N_i = \frac{1}{4} (1 - \eta^2) (1 + \zeta_0) (1 + \xi_0)$$

c) for $\zeta_i = 0$ $\xi_i = \pm 1$ and $\eta_i = \pm 1$

$$N_i = \frac{1}{4} (1 - \zeta^2) (1 + \xi_0) (1 + \eta_0)$$

(2.4)

Where $\xi_0 = \xi \xi_i$, $\eta_0 = \eta \eta_i$ and $\zeta_0 = \zeta \zeta_i$

in which ξ_i , η_i and ζ_i are the local natural coordinates of node i .

For example, for the element nodal configuration, as shown in Fig 2.2 the above expressions yield,

For corner node $i = 1$

$$\xi_i = -1, \quad \eta_i = -1 \quad \text{and} \quad \zeta_i = -1$$

$$N_1 = \frac{1}{8} (1 - \xi) (1 - \eta) (1 - \zeta) (-\xi - \eta - \zeta - 2)$$

For mid-side node $i = 15$

$$\xi_i = +1, \quad \eta_i = 0 \quad \text{and} \quad \zeta_i = -1$$

$$N_{15} = \frac{1}{4} (1 - \eta^2) (1 - \zeta) (1 + \xi)$$

The actual coordinates r , z and θ of a point within an element, prescribed in natural coordinates ξ , η and ζ , can be given by the following coordinate transformation relationships.

$$r = N_1 r_1 + N_2 r_2 + \dots + N_{20} r_{20}$$

$$z = N_1 z_1 + N_2 z_2 + \dots + N_{20} z_{20}$$

$$\theta = N_1 \theta_1 + N_2 \theta_2 + \dots + N_{20} \theta_{20}$$

.. .. (2.5)

Where N_1 to N_{20} are the shape functions as defined in Equation (2.4), and r_1 , z_1 and θ_1 to r_{20} , z_{20} and θ_{20} are the actual coordinates of the element nodes.

Similarly, the displacements u , v and w at any point prescribed by ξ , η and ζ , within an element are given by,

$$\begin{Bmatrix} u \\ v \\ w \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & 0 & N_2 & . & . & . & . & 0 & 0 \\ 0 & N_1 & 0 & 0 & . & . & . & . & N_{20} & 0 \\ 0 & 0 & N_1 & 0 & . & . & . & . & 0 & N_{20} \end{bmatrix} \cdot \begin{Bmatrix} u_1 \\ v_1 \\ w_1 \\ u_2 \\ . \\ . \\ . \\ v_{20} \\ w_{20} \end{Bmatrix} \quad \dots \quad (2.6)$$

N

where u_1, v_1 and w_1 to u_{20}, v_{20} and w_{20} are the element nodal displacements.

The differential operator matrix $[L]$ relating the strains and displacements at any point, viz

$$\begin{Bmatrix} \epsilon_r \\ \epsilon_z \\ \epsilon_\theta \\ \nu_{rz} \\ \nu_{r\theta} \\ \nu_{z\theta} \end{Bmatrix} = [L] \cdot \begin{Bmatrix} u \\ v \\ w \end{Bmatrix}$$

is obtained from Eqn (2.1) as,

$$[L] = \begin{bmatrix} \frac{\partial}{\partial r} & 0 & 0 \\ 0 & \frac{\partial}{\partial z} & 0 \\ \frac{1}{r} & 0 & \frac{1}{r} \frac{\partial}{\partial \theta} \\ \frac{\partial}{\partial z} & \frac{\partial}{\partial r} & 0 \\ \frac{1}{r} \frac{\partial}{\partial \theta} & 0 & \frac{\partial}{\partial r} - \frac{1}{r} \\ 0 & \frac{1}{r} \frac{\partial}{\partial \theta} & \frac{\partial}{\partial z} \end{bmatrix} \quad \dots \quad (2.7)$$

The strain matrix $[B]$ is then obtained as

$$[B] = [L] \cdot [N] \text{ from Eqns (2.6) \& (2.7)}$$

$$\text{or } [B]_i = [L] [N]_i \text{ for } i = 1, 20$$

$$\text{and } [B]_i = \begin{bmatrix} \frac{\partial N_i}{\partial r} & 0 & 0 \\ 0 & \frac{\partial N_i}{\partial z} & 0 \\ \frac{N_i}{r} & 0 & \frac{1}{r} \frac{\partial N_i}{\partial \theta} \\ \frac{\partial N_i}{\partial z} & \frac{\partial N_i}{\partial r} & 0 \\ \frac{1}{r} \frac{\partial N_i}{\partial \theta} & 0 & \frac{\partial N_i}{\partial r} - \frac{N_i}{r} \\ 0 & \frac{1}{r} \frac{\partial N_i}{\partial \theta} & \frac{\partial N_i}{\partial z} \end{bmatrix} \dots \dots (2.8)$$

The partial derivatives $\frac{\partial N_i}{\partial r}$, $\frac{\partial N_i}{\partial z}$ and $\frac{\partial N_i}{\partial \theta}$ cannot be evaluated directly as N_i are defined in ξ , η and ζ . They are obtained as follows.

We can write,

$$\frac{\partial N_i}{\partial \xi} = \frac{\partial N_i}{\partial r} \cdot \frac{\partial r}{\partial \xi} + \frac{\partial N_i}{\partial z} \cdot \frac{\partial z}{\partial \xi} + \frac{\partial N_i}{\partial \theta} \cdot \frac{\partial \theta}{\partial \xi},$$

$$\frac{\partial N_i}{\partial \eta} = \frac{\partial N_i}{\partial r} \cdot \frac{\partial r}{\partial \eta} + \frac{\partial N_i}{\partial z} \cdot \frac{\partial z}{\partial \eta} + \frac{\partial N_i}{\partial \theta} \cdot \frac{\partial \theta}{\partial \eta} \text{ and}$$

$$\frac{\partial N_i}{\partial \zeta} = \frac{\partial N_i}{\partial r} \cdot \frac{\partial r}{\partial \zeta} + \frac{\partial N_i}{\partial z} \cdot \frac{\partial z}{\partial \zeta} + \frac{\partial N_i}{\partial \theta} \cdot \frac{\partial \theta}{\partial \zeta}$$

$$\text{or } \begin{Bmatrix} \frac{\partial N_i}{\partial \xi} \\ \frac{\partial N_i}{\partial \eta} \\ \frac{\partial N_i}{\partial \zeta} \end{Bmatrix} = [J] \begin{Bmatrix} \frac{\partial N_i}{\partial r} \\ \frac{\partial N_i}{\partial z} \\ \frac{\partial N_i}{\partial \theta} \end{Bmatrix} \dots \dots (2.9)$$

where,

$$[J] = \begin{bmatrix} \frac{\partial r}{\partial \xi} & \frac{\partial z}{\partial \xi} & \frac{\partial \theta}{\partial \xi} \\ \frac{\partial r}{\partial \eta} & \frac{\partial z}{\partial \eta} & \frac{\partial \theta}{\partial \eta} \\ \frac{\partial r}{\partial \zeta} & \frac{\partial z}{\partial \zeta} & \frac{\partial \theta}{\partial \zeta} \end{bmatrix}$$

Now, we can write,

$$\begin{Bmatrix} \frac{\partial N_i}{\partial r} \\ \frac{\partial N_i}{\partial z} \\ \frac{\partial N_i}{\partial \theta} \end{Bmatrix} = [J]^{-1} \cdot \begin{Bmatrix} \frac{\partial N_i}{\partial \xi} \\ \frac{\partial N_i}{\partial \eta} \\ \frac{\partial N_i}{\partial \zeta} \end{Bmatrix} \quad \dots \dots (2.10)$$

In the above, $[J]$ is called the Jacobian transformation matrix, and its elements are evaluated using the relationships given in Eqn (2.5). For example,

$$\frac{\partial r}{\partial \xi} = \sum \frac{\partial N_i}{\partial \xi} \cdot r_i \quad \text{for } i = 1, 20$$

The stiffness matrix $[K]$ of an element is obtained from the following volume integral,

$$[K]_{60 \times 60} = \int_{vol} [B]_{60 \times 6}^T [D]_{6 \times 6} [B]_{6 \times 60} dv \quad \dots \dots (2.11)$$

The evaluation of the above integral requires the use of a numerical integration process, which is elaborated in a later section. The calculation of the stiffness product $B^T D B$, is easier to program in the above form as it stands, but would result in some waste computation due to the presence of zero coefficients in the $[B]$ and $[D]$ matrices. It is found that as much as 50% saving in element stiffness matrix computation can be achieved (Ref 30) by programming the explicit triple product of $B^T D B$, obtained for a pair of nodes i and j as described below.

From Eqn (2.11), the submatrix K_{ij} , of size (3×3) , relating to a pair of nodes i and j can be written as,

$$[K_{ij}]_{3 \times 3} = \int_{vol} [B_i]^T_{3 \times 6} \cdot [D]_{6 \times 6} \cdot [B_j]_{6 \times 3} \cdot dv \quad \dots \quad (2.12)$$

where i and j varies over node numbers from 1 to 20.

The Stress matrix $[Q_j] = [D][B_j]$ for any node j is given using Eqns (2.3) and (2.8) by,

$$[Q_j]_{6 \times 3} = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} \begin{bmatrix} (D_1 \frac{\partial N_j}{\partial r} + D_2 \frac{N_j}{r}) & D_2 \frac{\partial N_j}{\partial z} & D_2 \frac{1}{r} \frac{\partial N_j}{\partial \theta} \\ (D_2 \frac{\partial N_j}{\partial r} + D_2 \frac{N_j}{r}) & D_1 \frac{\partial N_j}{\partial z} & D_2 \frac{1}{r} \frac{\partial N_j}{\partial \theta} \\ (D_2 \frac{\partial N_j}{\partial r} + D_1 \frac{N_j}{r}) & D_2 \frac{\partial N_j}{\partial z} & D_1 \frac{1}{r} \frac{\partial N_j}{\partial \theta} \\ D_3 \frac{\partial N_j}{\partial z} & D_3 \frac{\partial N_j}{\partial r} & 0 \\ D_3 \frac{1}{r} \frac{\partial N_j}{\partial \theta} & 0 & D_3 (\frac{\partial N_j}{\partial r} - \frac{N_j}{r}) \\ 0 & D_3 \frac{1}{r} \frac{\partial N_j}{\partial \theta} & D_3 \frac{\partial N_j}{\partial z} \end{bmatrix} \quad \dots \quad (2.13)$$

where $D_1 = 1$, $D_2 = \frac{\nu}{1-\nu}$ and $D_3 = \frac{1-2\nu}{2(1-\nu)}$

Further, the triple product matrix $[K_{ij}]$ of Eqn (2.12) is given by,

$$[K_{ij}] = \int_{vol} [B_i]^T [Q_j] dv$$

or,

$$\begin{aligned}
 [K_{ij}]_{3 \times 3} &= \int_{vol} \left[\begin{array}{ccc}
 \frac{\partial N_i}{\partial r} Q_{11} + \frac{N_i}{r} Q_{31} & \frac{\partial N_i}{\partial r} Q_{12} + \frac{N_i}{r} Q_{32} & \frac{\partial N_i}{\partial r} Q_{13} + \frac{N_i}{r} Q_{33} \\
 + \frac{\partial N_i}{\partial z} Q_{41} + \frac{1}{r} \frac{\partial N_i}{\partial \theta} Q_{51} & + \frac{\partial N_i}{\partial z} Q_{42} & + \frac{1}{r} \frac{\partial N_i}{\partial \theta} Q_{53} \\
 \\
 \frac{\partial N_i}{\partial z} Q_{21} + \frac{\partial N_i}{\partial r} Q_{41} & \frac{\partial N_i}{\partial z} Q_{22} + \frac{\partial N_i}{\partial r} Q_{42} & \frac{\partial N_i}{\partial z} Q_{23} \\
 + \frac{1}{r} \frac{\partial N_i}{\partial \theta} Q_{62} & & + \frac{1}{r} \frac{\partial N_i}{\partial \theta} Q_{63} \\
 \\
 \frac{1}{r} \frac{\partial N_i}{\partial \theta} Q_{31} + & \frac{1}{r} \frac{\partial N_i}{\partial \theta} Q_{32} & \frac{1}{r} \frac{\partial N_i}{\partial \theta} Q_{33} \\
 (\frac{\partial N_i}{\partial r} - \frac{N_i}{r}) Q_{51} & + \frac{\partial N_i}{\partial z} Q_{62} & + (\frac{\partial N_i}{\partial r} - \frac{N_i}{r}) Q_{53} \\
 & & + \frac{\partial N_i}{\partial z} Q_{63}
 \end{array} \right] \times dV
 \end{aligned}
 \quad (2.14)$$

where $Q_{11}, Q_{12}, \dots$ etc. are the elements of the $[Q_j]$ as defined in Eqn (2.13); and the partial derivatives $\frac{\partial N_i}{\partial r}$, $\frac{\partial N_i}{\partial z}$ and $\frac{\partial N_i}{\partial \theta}$ are obtained using Eqn (2.10), where they are related to the local derivatives $\frac{\partial N_i}{\partial \xi}$, $\frac{\partial N_i}{\partial \eta}$ and $\frac{\partial N_i}{\partial \zeta}$

The local derivatives of the Shape functions N_i , as given in Eqn (2.4), are as shown below.

Corner nodes

$$\frac{\partial N_i}{\partial \xi} = \frac{1}{8} (1 + \eta_0) (1 + \zeta_0) (2\xi_0 + \eta_0 + \zeta_0 - 1) \xi_i$$

$$\frac{\partial N_i}{\partial \eta} = \frac{1}{8} (1 + \zeta_0) (1 + \xi_0) (2\eta_0 + \zeta_0 + \xi_0 - 1) \eta_i$$

$$\frac{\partial N_i}{\partial \zeta} = \frac{1}{8} (1 + \xi_0) (1 + \eta_0) (2\zeta_0 + \xi_0 + \eta_0 - 1) \zeta_i$$

Mid-Side nodes

$$a) \quad \xi_i = 0 \quad \eta_i = \pm 1 \quad \zeta_i = \pm 1$$

$$\frac{\partial N_i}{\partial \xi} = -\frac{1}{2} \xi (1 + \eta_0)(1 + \zeta_0)$$

$$\frac{\partial N_i}{\partial \eta} = \frac{1}{4} (1 - \xi^2)(1 + \zeta_0) \eta_i$$

$$\frac{\partial N_i}{\partial \zeta} = \frac{1}{4} (1 - \xi^2)(1 + \eta_0) \zeta_i$$

$$b) \quad \eta_i = 0 \quad \xi_i = \pm 1 \quad \zeta_i = \pm 1$$

$$\frac{\partial N_i}{\partial \xi} = \frac{1}{4} (1 - \eta^2)(1 + \zeta_0) \xi_i$$

$$\frac{\partial N_i}{\partial \eta} = -\frac{1}{2} \eta (1 + \zeta_0)(1 + \xi_0)$$

$$\frac{\partial N_i}{\partial \zeta} = \frac{1}{4} (1 - \eta^2)(1 + \xi_0) \zeta_i$$

$$c) \quad \zeta_i = 0 \quad \xi_i = \pm 1 \quad \eta_i = \pm 1$$

$$\frac{\partial N_i}{\partial \xi} = \frac{1}{4} (1 - \zeta^2)(1 + \eta_0) \xi_i$$

$$\frac{\partial N_i}{\partial \eta} = \frac{1}{4} (1 - \zeta^2)(1 + \xi_0) \eta_i$$

$$\frac{\partial N_i}{\partial \zeta} = -\frac{1}{2} \zeta (1 + \xi_0)(1 + \eta_0) \quad \dots \quad (2.15)$$

The computation of element stiffness matrices can be made more efficient, by use of an alternative formulation (Ref 37), for problems which have isotropic material properties which are constant within an element. This formulation is based on the internal energy concept and

uses Lamé's constants in place of elastic constants for the material. However, since the formulation proposed by reference 37 is only valid for isotropic materials, in order to keep generality of the formulation, it has not been used in the current analysis.

2.5 15-NODE ISOPARAMETRIC PRISM ELEMENT

The fifteen-node prism element in the local natural coordinate system and a representative mapped element in the actual coordinate system is shown in Fig 2.3.

The shape functions for the element (Ref 30) are given as follows:

$$\begin{aligned}
 N_1 &= \frac{1}{2} L_1(2L_1 - 1)(1 + \zeta) - \frac{1}{2} L_1(1 - \zeta^2) \\
 N_2 &= \frac{1}{2} L_2(2L_2 - 1)(1 + \zeta) - \frac{1}{2} L_2(1 - \zeta^2) \\
 N_3 &= \frac{1}{2} L_3(2L_3 - 1)(1 + \zeta) - \frac{1}{2} L_3(1 - \zeta^2) \\
 N_4 &= \frac{1}{2} L_1(2L_1 - 1)(1 - \zeta) - \frac{1}{2} L_1(1 - \zeta^2) \\
 N_5 &= \frac{1}{2} L_2(2L_2 - 1)(1 - \zeta) - \frac{1}{2} L_2(1 - \zeta^2) \\
 N_6 &= \frac{1}{2} L_3(2L_3 - 1)(1 - \zeta) - \frac{1}{2} L_3(1 - \zeta^2) \\
 N_7 &= L_1(1 - \zeta^2) \\
 N_8 &= L_2(1 - \zeta^2) \\
 N_9 &= L_3(1 - \zeta^2) \\
 N_{10} &= 2L_1L_2(1 + \zeta) \\
 N_{11} &= 2L_2L_3(1 + \zeta) \\
 N_{12} &= 2L_3L_1(1 + \zeta) \\
 N_{13} &= 2L_1L_2(1 - \zeta) \\
 N_{14} &= 2L_2L_3(1 - \zeta) \\
 N_{15} &= 2L_3L_1(1 - \zeta)
 \end{aligned}
 \quad \dots \quad (2.16)$$

where L_1 , L_2 and L_3 are the natural coordinates, for the triangular cross-section, described as area-coordinates, and ζ is the natural coordinate along the length of the prism.

The procedure employed to evaluate the element stiffness matrix $[K]$ is identical to that described previously for the 20-node element. All the relations from Eqn (2.5) to Eqn. (2.14) are valid, with the obvious change in number of nodes to 15, in place of 20. The resulting element stiffness matrix is of size (45 x 45).

However, the evaluation of derivatives $\frac{\partial N_i}{\partial r}$, $\frac{\partial N_i}{\partial z}$ and $\frac{\partial N_i}{\partial \theta}$ as described in Eqn (2.10) is not straight-forward in this case as the local coordinates L_1 , L_2 , L_3 and ζ are one more in number, resulting in a rectangular Jacobian matrix $[J]$ whose inverse is sought. But, with the following mathematical manipulation the situation is overcome.

Using the inter-relation of area-coordinates, $L_1 + L_2 + L_3 = 1$, a new set of fictitious local coordinates ξ and η are defined by,

$$\begin{aligned}\xi &= L_1 \\ \eta &= L_2 \quad \text{and} \\ 1 - \xi - \eta &= L_3\end{aligned}\quad \dots \dots (2.17)$$

It is now possible to write

$$\begin{aligned}\frac{\partial N_i}{\partial \xi} &= \frac{\partial N_i}{\partial L_1} \cdot \frac{\partial L_1}{\partial \xi} + \frac{\partial N_i}{\partial L_2} \cdot \frac{\partial L_2}{\partial \xi} + \frac{\partial N_i}{\partial L_3} \cdot \frac{\partial L_3}{\partial \xi} \\ &+ \frac{\partial N_i}{\partial L_1} - \frac{\partial N_i}{\partial L_3}, \quad \text{and similarly} \\ \frac{\partial N_i}{\partial \eta} &= \frac{\partial N_i}{\partial L_2} - \frac{\partial N_i}{\partial L_3}\end{aligned}\quad \dots \dots (2.18)$$

With the derivatives $\frac{\partial N_i}{\partial \xi}$ and $\frac{\partial N_i}{\partial \eta}$ available, the relation given in Eqn (2.10) is now applicable to the 15-node element.

The local derivatives of the shape functions $\frac{\partial N_i}{\partial \xi}$, $\frac{\partial N_i}{\partial \eta}$ and $\frac{\partial N_i}{\partial \zeta}$ obtained using the above relationships, are listed in Table 2.1.

2.6. LOAD VECTOR DUE TO ROTATION

The centrifugal body forces developed in an element due to the rotation of an impeller, are represented in finite element formulation as follows.

If b_r , b_z and b_θ are the components of body force per unit volume in the radial, axial and tangential directions respectively, the equivalent load vector is described by,

Table 2.1

Node No.	$\frac{\partial \eta_i}{\partial \xi}$	$\frac{\partial \eta_i}{\partial \eta}$	$\frac{\partial \eta_i}{\partial \zeta}$
1	$\frac{1}{2}(1+\zeta)(4L_1-1) - \frac{1}{2}(1-\zeta^2)$	0	$\frac{1}{2}L_1(2L_1-1) + L_1\zeta$
2	0	$\frac{1}{2}(1+\zeta)(4L_2-1) - \frac{1}{2}(1-\zeta^2)$	$\frac{1}{2}L_2(2L_2-1) + L_2\zeta$
3	$-\frac{1}{2}(1+\zeta)(4L_3-1) - \frac{1}{2}(1-\zeta^2)$	$-\frac{1}{2}(1+\zeta)(4L_3-1) - \frac{1}{2}(1-\zeta^2)$	$\frac{1}{2}L_3(2L_3-1) + L_3\zeta$
4	$\frac{1}{2}(1-\zeta)(4L_1-1) - \frac{1}{2}(1-\zeta^2)$	0	$-\frac{1}{2}L_1(2L_1-1) + L_1\zeta$
5	0	$\frac{1}{2}(1-\zeta)(4L_2-1) - \frac{1}{2}(1-\zeta^2)$	$-\frac{1}{2}L_2(2L_2-1) + L_2\zeta$
6	$-\frac{1}{2}(1-\zeta)(4L_3-1) - \frac{1}{2}(1-\zeta^2)$	$-\frac{1}{2}(1-\zeta)(4L_3-1) - \frac{1}{2}(1-\zeta^2)$	$-\frac{1}{2}L_3(2L_3-1) + L_3\zeta$
7	$1 - \zeta^2$	0	$-2L_1\zeta$
8	0	$1 - \zeta^2$	$-2L_2\zeta$
9	$-(1-\zeta^2)$	$-(1-\zeta^2)$	$-2L_3\zeta$
10	$2L_2(1+\zeta)$	$2L_1(1+\zeta)$	$2L_1L_2$
11	$-2L_2(1+\zeta)$	$2(1+\zeta)(L_3-L_2)$	$2L_2L_3$
12	$2(1+\zeta)(L_3-L_1)$	$-2L_1(1+\zeta)$	$2L_3L_1$
13	$2L_2(1-\zeta)$	$2L_1(1-\zeta)$	$-2L_1L_2$
14	$-2L_2(1-\zeta)$	$2(1-\zeta)(L_3-L_2)$	$-2L_2L_3$
15	$2(1-\zeta)(L_3-L_1)$	$-2L_1(1-\zeta)$	$-2L_3L_1$

$$\{f\} = \int_{vol} [N]^T \{b\} dv \quad \dots \quad (2.19)$$

$$\text{where } \{b\} = \begin{Bmatrix} b_r \\ b_z \\ b_\theta \end{Bmatrix}$$

For the case of centrifugal loading,

$$b_r = \rho \omega^2 r, \quad b_z = 0 \quad \text{and} \quad b_\theta = 0 \quad \dots \quad (2.20)$$

where ρ is the mass density of the material

ω is the angular velocity in rad/sec

r is the radius.

With this, the load vector for 20-node element is

$$\{f\} = \int_{vol} \begin{bmatrix} N_1 & 0 & 0 \\ 0 & N_1 & 0 \\ 0 & 0 & N_1 \\ \vdots & \vdots & \vdots \\ \vdots & \vdots & \vdots \\ \vdots & \vdots & \vdots \\ N_{20} & 0 & 0 \\ 0 & N_{20} & 0 \\ 0 & 0 & N_{20} \end{bmatrix} \begin{Bmatrix} b_r \\ 0 \\ 0 \end{Bmatrix} dV$$

or

$$\text{or } \{f\} = \int_{\text{vol}} \begin{Bmatrix} N_1 b_r \\ 0 \\ 0 \\ N_2 b_r \\ \vdots \\ \vdots \\ \vdots \\ N_{20} b_r \\ 0 \\ 0 \end{Bmatrix} \cdot dV \quad \dots \quad (2.21)$$

Similarly, the load vector for the 15-node element is,

$$\{f\} = \int_{\text{vol}} \begin{Bmatrix} N_1 b_r \\ 0 \\ 0 \\ N_2 b_r \\ \vdots \\ \vdots \\ \vdots \\ N_{15} b_r \\ 0 \\ 0 \end{Bmatrix} dV \quad \dots \quad (2.22)$$

2.7 NUMERICAL INTEGRATION

The explicit evaluation of the volume integrals encountered in the calculation of stiffness matrices and load vectors is unrealistic due to the complexity of the expressions involved, and numerical integration is usually employed in order to obtain a solution. The numerical integration, while necessary, also helps in simplifying the computer implementation of element routines.

The volume integrals encountered, viz Eqns (2.11) and (2.19) are of the form $\int_V \phi dV$, where ϕ is a function of the local coordinates ξ , η and ζ , which take values between ± 1 . To evaluate these integrals the variables and the limits of integration need to be transformed as follows (Ref 32).

$$\begin{aligned} \int_V \phi(\xi, \eta, \zeta) dV &= \iiint \phi(\xi, \eta, \zeta) dr dz r d\theta \\ &= \int_{-1}^{+1} \int_{-1}^{+1} \int_{-1}^{+1} \phi(\xi, \eta, \zeta) \frac{\partial(r, z, \theta)}{\partial(\xi, \eta, \zeta)} r d\xi d\eta d\zeta \quad \dots \quad (2.23) \end{aligned}$$

$$\text{and } \frac{\partial(r, z, \theta)}{\partial(\xi, \eta, \zeta)} = \begin{vmatrix} \frac{\partial r}{\partial \xi} & \frac{\partial z}{\partial \xi} & \frac{\partial \theta}{\partial \xi} \\ \frac{\partial r}{\partial \eta} & \frac{\partial z}{\partial \eta} & \frac{\partial \theta}{\partial \eta} \\ \frac{\partial r}{\partial \zeta} & \frac{\partial z}{\partial \zeta} & \frac{\partial \theta}{\partial \zeta} \end{vmatrix} \quad \dots \quad (2.24)$$

The above is the determinant of the Jacobian transformation matrix as described in Eqn (2.9). Eqn (2.23) can be written as,

$$\int_{vol} \phi(\xi, \eta, \zeta) dV = \int_{-1}^{+1} \int_{-1}^{+1} \int_{-1}^{+1} \phi(\xi, \eta, \zeta) \det[J] r d\xi d\eta d\zeta \quad \dots \quad (2.25)$$

Gauss Quadrature

The Gauss Quadrature formulae (Ref 30) are ideally suited for the evaluation of integrals of the type in Eqn (2.25), and they give increased accuracy for a given number of sampling points.

The basic formula for a line integral takes the form

$$\int_{-1}^{+1} f(\xi) d\xi = \sum_{i=1}^n H_i f(\xi_i) \quad \dots \quad (2.26)$$

where n is the number of sampling points
 ξ_i specifies the position of sampling (integration) point
 and H_i is the corresponding weight coefficient

The above formula can be extended to integrate the volume of a light prism as,

$$\int_{-1}^{+1} \int_{-1}^{+1} \int_{-1}^{+1} f(\xi, \eta, \zeta) d\xi d\eta d\zeta = \sum_{m=1}^n \sum_{j=1}^n \sum_{i=1}^n H_i H_j H_m f(\xi_i, \eta_j, \zeta_m) \quad \dots \quad (2.27)$$

where n is the number of sampling points in each direction (which can be different)
 ξ_i, η_j & ζ_m specifies the position of a sampling point in the light prism
 H_i, H_j & H_m are the corresponding weight coefficients

The abscissae and weight coefficients of Gauss Quadrature points for various orders of integration, of a line integral, are given in Table 8.1 of reference 30.

Triangular Regions

The integrals encountered in the evaluation of the stiffness matrices for triangular elements are of the form $\int_A \phi(L_1, L_2, L_3) dA$ as the shape functions are defined in area coordinates L_1, L_2 and L_3 . The Gauss Quadrature described earlier is not directly applicable, however sampling points defined in area coordinates are used in a similar fashion.

As the area coordinates are inter-related by the expression $L_1 + L_2 + L_3 = 1$, one could define L_3 in terms of L_1 and L_2 . Thus the integration need to be carried out with respect to only two independent variables L_1 and L_2 .

As in Eqn (2.17), if we define $\xi = L_1$, $\eta = L_2$ and $1-\xi-\eta = L_3$, we can write for a triangle defined in the r - z plane as,

$$\int_{\text{Area}} \phi(L_1, L_2, L_3) dr dz = \frac{1}{2} \int_0^1 \int_0^{1-\xi} \phi(L_1, L_2, L_3) \frac{\partial(r, z)}{\partial(\xi, \eta)} d\eta d\xi \quad (2.28)$$

$$\text{and } \frac{\partial(r, z)}{\partial(\xi, \eta)} = \begin{vmatrix} \frac{\partial r}{\partial \xi} & \frac{\partial z}{\partial \xi} \\ \frac{\partial r}{\partial \eta} & \frac{\partial z}{\partial \eta} \end{vmatrix} \quad (2.29)$$

which is the Jacobian determinant.

To evaluate the above integrals, an expression similar to the Gauss Quadrature is given as follows (Ref 30),

$$\int_0^1 \int_0^{1-L_1} f(L_1, L_2, L_3) dL_2 dL_1 = \sum_{i=1}^n H_i f_i(L_1, L_2, L_3) \quad (2.30)$$

where n number of sampling points in the triangle

f_i position of the sampling point

H_i the corresponding weight coefficient.

It is interesting to note that the summation expression in the above equation does not give any bias to any of the natural coordinates L_1 , unlike the integral on the left hand side.

A series of sampling points and weights, for various orders of integration, are given in reference 38, and are reproduced in Table 8.2 of reference 30. During the current investigation, a new set of integration points for triangles, suiting Eqn (2.30), and evolved from the basic Gauss Quadrature points, are identified (Ref 39).

Triangular Prisms

From the view-point of integration, triangular prisms are hybrid in that they possess a triangular cross-section with a longitudinal axis, as shown previously in Fig 2.3. The type of volume integrals encountered in the stiffness expressions are of the form

$$\int_{vol} \phi (L_1, L_2, L_3, \zeta) dV$$

As in Eqns (2.25) and (2.28) the integrals can be written as

$$\int_{vol} \phi (L_1, L_2, L_3, \zeta) dV = \frac{1}{2} \int_{-1}^{+1} \int_0^1 \int_0^{1-\xi} \phi (L_1, L_2, L_3, \zeta) \det|J| d\eta d\xi d\zeta \quad \dots \dots (2.31)$$

where $\xi = L_1$, $\eta = L_2$ and $1-\xi-\eta = L_3$

By combining the integration formulae of Eqns (2.26) and (2.30) it is possible to write

$$\int_{-1}^{+1} \int_0^1 \int_0^{1-L_1} f (L_1, L_2, L_3, \zeta) dL_2 dL_1 d\zeta = \sum_{i=1}^{n_1} \sum_{j=1}^{n_2} H_i H_j f((L_1, L_2, L_3)_i, \zeta_j) \quad \dots \dots (2.32)$$

where n_1 = number of sampling points in the triangular cross section
 n_2 = number of sampling points in the longitudinal direction

Minimum Integration

With numerical integration employed in place of an exact solution, an additional error is introduced into the calculation. At first sight it would appear that this should be reduced as much as possible by the use of a higher order of integration. Early generation programs used a comparable amount of computer time for the generation of elements as for the subsequent solution of equations. Later research has proved that it is often a positive disadvantage to use higher orders of integration than those actually needed and necessary just to preserve the rate of convergence which would result if exact integration were used (Ref 30). For a number of reasons, the cancellation of errors due to discretization and inexact integration occurs, thus improving the quality

of the results obtained. The improvement so obtained is significant particularly for the case of near incompressible materials, with Poisson's ratios near 0.5. In such cases the higher order integrations fail to produce any meaningful results.

The integration points chosen, for the two elements considered, are as follows.

20-node Element

The evaluation of volume integrals involved in the calculation of the stiffness properties, is carried out with the use of the Gauss Quadrature formula as described in Eqn (2.27). The $2 \times 2 \times 2$ Gauss points, which are recommended for minimum integration (Ref 30), are used. At this point it is worth mentioning the difficulties experienced with these points at the early stages of the current investigation.

The simple classical problem of a ring under centrifugal pull was considered, to prove the element routine. A sector of the ring was treated as a repeatable segment and was modelled with three 20-node elements joined along the length of the sector. With a normal solution, the displacements obtained were such that the average radial growth agreed with the classical solution, but parabolic distortions were observed on the sides of the sector cross section as shown in Fig 2.4.

With diagonal decay criteria incorporated in the solution (Refs 31, 40) the program failed during Gaussian elimination, showing extreme diagonal decay due to round-off damage. This failure indicated some ill-conditioning of the stiffness matrix of the structure. Further, when the $2 \times 2 \times 2$ rule was replaced by a $3 \times 3 \times 3$ rule or 14-point rule (Refs 41, 42) for integration, satisfactory solutions were produced. It was thus concluded that this particular form of structure falls into a special category wherein the minimum integration procedure leaves room for mechanisms, even though it does satisfy the broad rule laid down in reference 30 "that the number of independent relations supplied at all the integration points should be more than the total number of unknowns (displacements) for a non-singular stiffness matrix". In a recent publication (Ref 43), Irons confirmed this experience and explained the phenomena under the heading "Mechanisms".

The $2 \times 2 \times 2$ rule gave no difficulties such as those described above when used for the case of a complete segment of an impeller. In fact, the results showed improvement over those obtained using a higher order of integration, confirming the earlier research on "minimum integration". It gave excellent results even for the case of materials with Poisson's ratios very near 0.5. In view of these satisfactory results, the $2 \times 2 \times 2$ rule is adopted as a final choice.

15-node Element

The evaluation of volume integrals is carried out using the expression in Eqn (2.32). The Gauss Quadrature 2-point formula (along the axis of the prism) and the 3-point formula for quadratic integration of triangles are combined to produce a 2 x 3 rule for the integration of triangular prisms.

2.8 THE PRINCIPLE OF REPEATABILITY

It is well appreciated that symmetry of a structure can permit a considerable reduction in the size of the problem, achieving significant economies in the cost of analysis. However, in numerous cases a repetition of structural form and loading is present although no axis of symmetry exists. In such cases, similar economies can be achieved by the use of the principle of repeatability (Ref 29).

Consider the case of an infinite blade cascade (Fig 2.5(a)). It is evident that each segment behaves identically to the next one, and thus such functions as displacements and stresses are identical for each segment.

Isolating a typical segment between sections AA and BB (Fig 2.5(b)) for analysis purpose a stiffness relationship can be written as follows:

$$\begin{bmatrix} K_{II} & K_{IA} & K_{IB} \\ & K_{AA} & K_{AB} \\ \text{SYM} & & K_{BB} \end{bmatrix} \cdot \begin{Bmatrix} \delta_I \\ \delta_A \\ \delta_B \end{Bmatrix} = \begin{Bmatrix} F_I \\ F_A \\ F_B \end{Bmatrix} + \begin{Bmatrix} 0 \\ R_A \\ R_B \end{Bmatrix} \quad \dots \quad (2.33)$$

where $\{\delta\}$ lists the displacements, $\{F\}$ the active forces and $\{R\}$ the reactions from the removed structure. The subscripts I, A and B represent the nodes present internally, on section AA and on section BB respectively.

As all the segments are identical, it is evident that

$$\{\delta_A\} = \{\delta_B\} \quad \text{and} \quad \{R_A\} = -\{R_B\} \quad \dots \quad (2.34)$$

Using the above relations, Eqn (2.33) can be rewritten as

$$\begin{bmatrix} K_{II} & K_{IA} + K_{IB} \\ & K_{AA} + K_{AB} + K_{BA} + K_{BB} \\ \text{SYM} & \end{bmatrix} \begin{Bmatrix} \delta_I \\ \delta_A \end{Bmatrix} = \begin{Bmatrix} F_I \\ F_A + F_B \end{Bmatrix} \quad \dots \quad (2.35)$$

The result is significant and can be achieved directly by the usual process of structural assembly by identifying the nodes on sections AA and BB a priori.

A principle of repeatability can be stated as follows, "if a system composed of a series of segments identical in structure and loading is considered it is only necessary to perform analysis on one typical segment identifying the nodes on the two sections which isolate it from the rest of the system".

In Fig 2.5(a), an alternative repeatable segment is shown between sections CC and DD. Thus what constitutes a repeatable segment is completely arbitrary. For cases of rotational repeatability, such as occur in the analysis of centrifugal impellers (Fig 2.5(c)), repeatable sectors are considered. This is only possible if they are treated in a cylindrical coordinate system.

2.9 ASSEMBLY AND SOLUTION

The stiffness relation for the complete structure takes the general form

$$[K] \{\delta\} = \{f\} + \{R\} \quad \dots \quad (2.36)$$

where $[K]$ = the stiffness matrix of the structure
 $\{\delta\}$ = the vector of displacements
 $\{f\}$ = the vector of loads
 and $\{R\}$ = the vector of reactions

The load vector in its general form is a sum of one or more type of loads acting on the structure, and is comprised of induced body forces, surface forces, forces developed due to initial strain, and forces developed due to the presence of any residual stresses.

The structure stiffness matrix and the load vector are assembled using the relations

$$K_{ij} = \sum K_{ij}^e \quad \text{and} \quad f_i = \sum f_i^e \quad \dots \quad (2.37)$$

where K_{ij}^e and f_i^e are the corresponding element coefficients, with a sum taken over all the elements in which they occur.

Boundary Conditions

For a structure with n number of degrees of freedom, the matrix Eqn (2.36) will result in a set of n simultaneous equations and can be

solved for n unknowns comprised of displacements and reactions. If a structure is constrained at the points of constraint, the reactions are unknown, whereas the corresponding displacements are known (zero displacements). Elsewhere, the displacements are unknown with known zero reactions. In all such cases, the equations corresponding to the known displacements are deleted from the matrix Eqn (2.36), and the rest of the simultaneous equations are solved for the unknown displacements. However, in practice, more convenient numerical methods which can prescribe the displacements without the need to delete the corresponding equations are available. In situations where structures develop no external reactions, all the displacements are unknown. One such case being an impeller under rotation. In such cases a solution of the equations will be singular unless the rigid body displacements are suppressed. In fact, the structure stiffness matrix $[K]$ in its full original form is singular and possesses no inverse. To suppress the rigid body displacements a priori, it is necessary to identify them for the structure together with the considered finite element idealisation, and then to constrain a certain minimum number of degrees of freedom corresponding to the rigid body modes. The choice is arbitrary, but it is important that care is taken to avoid any introduction of additional strains in the structure. For the case of an impeller, modelled with solid elements, two rigid body modes are possible; one in the axial direction and the other in the circumferential direction. It is enough if two corresponding degrees of freedom are constrained at any one node in the structure, to remove the rigid body displacements.

Solution of Equations

More than one numerical method is available for the reduction of a set of simultaneous equations, with the Gaussian reduction being more commonly used. For large structures, it is impossible to store the complete stiffness matrix in the high speed core of a computer. All the solutions adopted for finite element analysis take advantage of the large number of zero coefficients present and go for some form of banded solution wherein mostly non-zero coefficients are stored in the computer. However, for the case of practical and three-dimensional structures the size of the matrix is so huge that even such solutions are inadequate. In ring type structures, the additional factor is the unavoidably large band-width which makes any banded solution inefficient. The alternative approach, which is very popular and ideally suited for finite element analysis, is the frontal solution.

Briefly, in a frontal solution, the assembly and reduction of the structure stiffness matrix and the load vector proceed simultaneously. At any given point of time, only a part of the matrix corresponding to the active variables is in high speed core. It is an element biased algorithm, as the assembly process is carried after the introduction of each element and the mature variables are reduced immediately after that. By a judicious choice of element sequence to the assembly, the high speed core requirements can be minimised. The frontal solution has added flexibilities such as the re-solution of new load vectors without the need for the reduction of the matrix all over again.

There are other major benefits occurring as a result of the frontal concept. One such advantage is that the nodal stress averages can be evaluated with minimal effort using the basic frontal logic. For more detail, indepth understanding and actual usage, the reader is referred to the original publication (Ref 31) on the subject. References 44 and 45 give more applications of the basic frontal concept in other areas such as dynamic analysis.

2.10 EVALUATION OF STRESSES

A most important and useful output from any finite element analysis is the stress distribution. As outlined in the preceding section, once the set of simultaneous equations relating the acting loads and displacements are solved all the displacements of the structure can be determined. In effect, the element nodal displacements for any element are available.

The displacements of any point within an element can be obtained using the Eqn (2.6). The strains and stresses at the point can then be calculated using the strain-displacement Eqn (2.1) and the stress-strain Eqn (2.2) respectively. Thus the stresses at any point within an element can be evaluated once the element nodal displacements are known.

Usually, stresses are produced at the nodes of a structure and they are computed as an average of the stresses obtained from the elements surrounding a node. Using the procedure described in the preceding paragraph, it is possible to calculate the stresses at the nodes of an element directly. However, the literature on the subject points out that such directly calculated stresses are relatively poor in accuracy (Ref 30). It is suggested that improved accuracy could be achieved by evaluating the stresses at the optimal sampling points, which are defined a priori, and then extrapolate the stresses at the nodes using these values. Such a process is called 'local least squares stress smoothing'. More details on the subject are available in reference 46.

In the following the optimal sampling points chosen for the two types of elements considered, together with the derivations of the stress smoothing matrices, are outlined.

20-node Element

In reference 47 the optimal sampling points are identified for this element, which happens to correspond to the minimum integrations points. The position of the eight sampling points within an element are marked in Fig 2.6.

The smoothed stress σ at any point within an element, may be described by a trilinear expression (Ref 48) as follows:

$$\sigma(\xi, \eta, \zeta) = a_1 + a_2 \xi + a_3 \eta + a_4 \zeta + a_5 \xi\eta + a_6 \xi\zeta + a_7 \eta\zeta + a_8 \xi\eta\zeta \quad \dots \quad (2.38)$$

where ξ , η and ζ are natural coordinates of any point within an element and a_1 to a_8 are the constants of the equation.

Using the above expression it is possible to write a set of eight simultaneous equations, corresponding to the eight sampling points, which are known and identified as i to viii. They are written in matrix form as,

$$\begin{Bmatrix} \sigma_i \\ \sigma_{ii} \\ \vdots \\ \sigma_{viii} \end{Bmatrix} = \begin{bmatrix} 1 & \xi_i & \eta_i & \zeta_i & \xi_i \eta_i & \xi_i \zeta_i & \eta_i \zeta_i & \xi_i \eta_i \zeta_i \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\ 1 & \xi_{viii} & \eta_{viii} & \zeta_{viii} & \xi_{viii} \eta_{viii} & \xi_{viii} \zeta_{viii} & \eta_{viii} \zeta_{viii} & \xi_{viii} \eta_{viii} \zeta_{viii} \end{bmatrix} \begin{Bmatrix} a_1 \\ a_2 \\ \vdots \\ a_8 \end{Bmatrix} \quad (2.39)$$

P

The constants a_1 to a_8 can be obtained as,

$$\begin{Bmatrix} a_1 \\ a_2 \\ \vdots \\ a_8 \end{Bmatrix} = [P]^{-1} \begin{Bmatrix} \sigma_i \\ \sigma_{ii} \\ \vdots \\ \sigma_{viii} \end{Bmatrix} \quad (2.40)$$

The smoothed stresses at the element nodes can be written using the Eqn (2.38) in matrix form as

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \vdots \\ \sigma_{20} \end{Bmatrix} = \begin{bmatrix} 1 & \xi_1 & \eta_1 & \zeta_1 & \xi_1 \eta_1 & \xi_1 \zeta_1 & \eta_1 \zeta_1 & \xi_1 \eta_1 \zeta_1 \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\ 1 & \xi_{20} & \eta_{20} & \zeta_{20} & \xi_{20} \eta_{20} & \xi_{20} \zeta_{20} & \eta_{20} \zeta_{20} & \xi_{20} \eta_{20} \zeta_{20} \end{bmatrix} \begin{Bmatrix} a_1 \\ a_2 \\ \vdots \\ a_8 \end{Bmatrix} \quad (2.41)$$

P'

Combining Eqns (2.40) and (2.41),

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \vdots \\ \sigma_{20} \end{Bmatrix} = [S] \cdot \begin{Bmatrix} \sigma_i \\ \sigma_{ii} \\ \vdots \\ \sigma_{viii} \end{Bmatrix} \quad \dots \quad (2.42)$$

$$\text{where } [S]_{20 \times 8} = [P']_{20 \times 8} \cdot [P]_{8 \times 8}^{-1} \quad \dots \quad (2.43)$$

The matrix $[S]$ which is called the smoothing matrix, relates the stresses at the element nodes to the stresses at the sampling points. As its coefficients are functions of only the natural coordinates (not the actual coordinates) of the nodes and sampling points, it is advisable to calculate it only once and store it, to avoid repetitive computation for the same.

15-node Element

The optimal sampling points for this element are not identified in the literature. However, if optimal sampling points for a 6-node quadratic triangle and 3-node quadratic line element are available, it is possible to combine them to get a set of points for this element, similar to the case for integration points. For a 3-node line element the optimal points are the same as the quadratic minimum integration points, defined by the 2-point Gauss Quadrature. But, for the case of a 6-node triangle, the optimal sampling points suggested in the literature (Ref 30), which are the four cubic integration points defined in area coordinates, are not convincing and at best are an arbitrary choice. At this point of the current investigation, trials were made to define optimal sampling points based on the locus of the Gauss Quadrature points. It resulted in a set of new integration points in area coordinates for the triangles and was reported separately in reference 39. The 3-point quadratic integration points thus obtained for triangles, were combined with 2-point Gauss Quadrature points, resulting in six sampling points within the prism which are shown in Fig 2.7. The same points can be used for minimum integration.

A trilinear expression may now be written to define the smoothed stress σ at any point within an element as follows:

$$\sigma(L_1, L_2, L_3, \zeta) = a_1 L_1 + a_2 L_2 + a_3 L_3 + a_4 L_1 \zeta + a_5 L_2 \zeta + a_6 L_3 \zeta \quad \dots \quad (2.44)$$

where L_1, L_2, L_3 and ζ are the natural coordinates of any point within an element and a_1 to a_6 are the constants of the equation.

Using the above expression and the coordinates of the six sampling points i to vi , a set of simultaneous equations in matrix form can be written as .

$$\begin{Bmatrix} \sigma_i \\ \sigma_{ii} \\ \vdots \\ \vdots \\ \vdots \\ \sigma_{vi} \end{Bmatrix} = \begin{bmatrix} L_1 & L_2 & L_3 & L_1 \zeta & L_2 \zeta & L_3 \zeta \\ i & i & i & i & i & i \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\ L_1 & L_2 & L_3 & L_1 \zeta & L_2 \zeta & L_3 \zeta \\ vi & vi & vi & vi & vi & vi \end{bmatrix} \begin{Bmatrix} a_1 \\ a_2 \\ \vdots \\ \vdots \\ \vdots \\ a_6 \end{Bmatrix} \quad \begin{matrix} \dots \\ \dots \\ \dots \\ \dots \\ \dots \\ \dots \end{matrix} \quad (2.45)$$

P

The constants a_1 to a_6 can be obtained as

$$\begin{Bmatrix} a_1 \\ a_2 \\ \vdots \\ \vdots \\ \vdots \\ a_6 \end{Bmatrix} = [P]^{-1} \begin{Bmatrix} \sigma_i \\ \sigma_{ii} \\ \vdots \\ \vdots \\ \vdots \\ \sigma_{vi} \end{Bmatrix} \quad \dots \quad (2.46)$$

The smoothed stresses at the element nodes can be written in the matrix form as

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \vdots \\ \vdots \\ \vdots \\ \sigma_{15} \end{Bmatrix} = \begin{bmatrix} L_1 & L_2 & L_3 & L_1 \zeta & L_2 \zeta & L_3 \zeta \\ 1 & 1 & 1 & 1 & 1 & 1 \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\ \vdots & \vdots & \vdots & \vdots & \vdots & \vdots \\ L_1 & L_2 & L_3 & L_1 \zeta & L_2 \zeta & L_3 \zeta \\ 15 & 15 & 15 & 15 & 15 & 15 \end{bmatrix} \begin{Bmatrix} a_1 \\ a_2 \\ \vdots \\ \vdots \\ \vdots \\ a_6 \end{Bmatrix} \quad \begin{matrix} \dots \\ \dots \\ \dots \\ \dots \\ \dots \\ \dots \end{matrix} \quad (2.47)$$

P'

Combining Eqns (2.46) and (2.47)

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \vdots \\ \sigma_{15} \end{Bmatrix} = [S] \cdot \begin{Bmatrix} \sigma_i \\ \sigma_{ii} \\ \vdots \\ \sigma_{vi} \end{Bmatrix} \quad \dots \quad (2.48)$$

$$\text{where } [S]_{15 \times 6} = [P]_{15 \times 6} [P]_{6 \times 6}^{-1} \quad \dots \quad (2.49)$$

Once again the smoothing matrix $[S]$ which relates the stresses at the element nodes to the sampling points, needs to be calculated only once for the element.

2.11 THERMAL LOADING

Structures can develop internal stresses when they are subjected to temperature gradients. As an example consider the case of a structure divided into fictitious three-dimensional solid elements which are subjected to a temperature distribution. Suppose all the elements are separated and allowed to expand freely subjected to their respective temperatures. For this case there will be no internal stresses developed in the elements. If these freely expanded elements were assembled, one would expect a misfit due to the differential growth of the elements. However, if the elements were now contained within the original geometry of the structure, the misfits would disappear resulting in the development of internal stresses.

In finite element analysis the above result is achieved exactly, but in a different manner. In the above, an element was allowed free expansion, before assembly. One could achieve the same expansion by an equivalent set of external forces, replacing the temperature rise. If such equivalent load vectors are determined for all the elements and are assembled, the resulting load vector for the structure would produce identical displacements of the structure, as the temperature distribution which it replaces. The so-called misfits during assembly, in the earlier explanation, are now taken care of analytically in the assembly of the structure load vector.

The strain at any point in the structure is the result of free thermal strain plus the strain due to differential growth. As free thermal expansion does not produce a stress, only the strain due to differential growth is to be considered for the calculation of the resulting stress. In the following, the appropriate matrix relations are presented.

As mentioned previously, the load vector in the stiffness Eqn (2.36) is of general form and can be due to initial strains in the structure. Thermal strain is a type of initial strain for the purpose of analysis.

The element load vector, due to the initial strain $\{\epsilon_0\}$ in an element, can be given as

$$\{f\} = \int_{vol} [B]^T [D] \{\epsilon_0\} dV \quad \dots \quad (2.50)$$

For the case of an isotropic material, the thermal strain will contain only normal strains and no shear strains, and it takes the form

$$\{\epsilon_0\} = \begin{Bmatrix} \epsilon_r \\ \epsilon_z \\ \epsilon_\theta \\ \epsilon_{rz} \\ \epsilon_{r\theta} \\ \epsilon_{z\theta} \end{Bmatrix} = \begin{Bmatrix} \alpha T \\ \alpha T \\ \alpha T \\ 0 \\ 0 \\ 0 \end{Bmatrix} \quad \dots \quad (2.51)$$

where α is the linear coefficient of thermal expansion for the material

and T is the temperature rise.

Substituting the Eqns (2.3), (2.8) and (2.51) the explicit triple product of Eqn (2.50), for any node i , can be written as follows,

$$\{f_i\} = \int_{vol} \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} (D_1 + 2D_2) \cdot \alpha T \cdot \begin{Bmatrix} \frac{\partial N_i}{\partial r} + \frac{N_i}{r} \\ \frac{\partial N_i}{\partial z} \\ \frac{1}{r} \frac{\partial N_i}{\partial \theta} \end{Bmatrix} dV \quad \dots \quad (2.52)$$

where $D_1 = 1$ and $D_2 = \frac{\nu}{1-\nu}$

The volume integrals in the above expression are evaluated using numerical integration, as described previously.

The temperature T at any point (integration point) within an element can be evaluated using a similar relation to that given in Eqn (2.5)

$$T = \sum N_i T_i \quad \dots \quad (2.53)$$

where N_i are the shape functions

and T_i are the nodal temperatures.

The assembly of element load vectors and the solution process is identical to that described previously for centrifugal loading.

As explained earlier, the stress at any point in the structure is related to the strain caused by differential growth, which is obtained as the actual strain minus the free thermal strain. This can be given as,

$$\{\sigma\} = [D] \{\epsilon - \epsilon_0\} \quad \dots \quad (2.54)$$

The rest of the procedure to calculate the stresses at the element nodes remains the same as that described earlier in Section 2.10.

2.12 FLUID PRESSURE LOADING

Unlike the problem of heat conduction within an impeller, the concept of loading due to fluid pressure is fairly straightforward. The analytical treatment is the same as that described for the case of centrifugal loading, except that the calculation of the load vector is now based upon the surface fluid pressure distribution. However, the calculation of the element load vector is now more complex than were the earlier cases. It now involves, not only the identification of element faces on which the fluid pressure is acting, but also surface integration of the faces which are now three-dimensional. In the earlier cases, the element load vector could be evaluated simultaneously with the element stiffness matrix in a sub-program, since both required the same volumetric numeric integration. However, for the present case, the element load vector must be evaluated separately from the stiffness matrix owing to the different surface integrations. While there is no straight-forward method to integrate the faces of a solid isoparametric element, which are in three-dimensional space, the method suggested by Irons (Ref 43) appears to be a simple and elegant process. However, vectoral thinking is required in order to understand the technique, which is described below.

Consider a brick element as shown in Fig 2.8, with covariant base vectors $\vec{\xi}$, $\vec{\eta}$, $\vec{\zeta}$. Recalling the Jacobian transformation matrix J , as given in equation (2.10), the first row now represents the above vector $\vec{\xi}$. Similarly, the second and third rows represent $\vec{\eta}$ and $\vec{\zeta}$ respectively.

Considering the face $\xi = 1$ for example, the elemental parallelogram has sides $\vec{\eta}d\eta$ and $\vec{\zeta}d\zeta$, and hence the vector area $d\vec{A}$ is given by

$$d\vec{A} = \vec{\eta} \times \vec{\zeta} d\eta d\zeta = \vec{A} d\eta d\zeta \quad \dots \quad (2.55)$$

If p is the pressure acting on the face, the nodal forces are given by

$$\{f\} = - \iint p [N]^T \{\vec{A}\} d\eta d\zeta \quad \dots \quad (2.56)$$

The terms of the area vector $\vec{A} = \vec{\eta} \times \vec{\zeta}$ can be extracted easily, by interpreting the inverse of the Jacobian matrix as follows.

$$[J]^{-1} = \frac{1}{\det J} \begin{bmatrix} \vec{\eta} \times \vec{\zeta} & \vec{\zeta} \times \vec{\xi} & \vec{\xi} \times \vec{\eta} \\ \downarrow & \downarrow & \downarrow \end{bmatrix}$$

The double integral in the above equation (2.56) is readily evaluated using the numerical integration described earlier in article 2.7.

If the pressure acting on an element is variable and is defined at the nodes of the element, the pressure intensity p at any point (integration point) can be evaluated using a similar relation as given in equation (2.5).

$$p = \sum N_i p_i \quad \dots \quad (2.57)$$

where N_i are the shape functions
and p_i are the nodal pressure intensities.

The only remaining problem lies in identifying the faces of an element in a program! One suggestion, for the program used in this

investigation, is to employ the following nomenclature for the six faces of a 20-node element (Fig 2.2).

Face 1	corresponds to	$\zeta = -1$	face
Face 2	"	$\zeta = +1$	face
Face 3	"	$\xi = -1$	face
Face 4	"	$\eta = +1$	face
Face 5	"	$\xi = +1$	face
Face 6	"	$\eta = -1$	face

CHAPTER 3

PROGRAMMING CONSIDERATIONS

3.1 GENERAL DESCRIPTION

Implementation of a three-dimensional finite element solution, for an industrial problem, usually involves a large computer capability. Developing a suitable computer program, which is efficient in terms of computer core and time requirements, is the primary task of any such analysis. Experience with large and complex programs emphasises the need for reliability, maintainability and extensibility of the programs. To incorporate these factors into a program, considerable organisation is required. Program readability is desirable as a consideration to the users who may wish to modify the programs, to suit their individual requirements.

The above factors were kept in mind whilst the algorithms described in the previous chapter were programmed. The ANSI version of FORTRAN IV has been generally adopted to maintain some compiler independence. However, some features of the extended FORTRAN, such as the use of free formats for input are retained, but are easily replaceable when required. Some of the general characteristics of the programs are described below.

3.2 GENERAL CHARACTERISTICS

The program package which was prepared is modular in structure in order to minimise the demand on computer core. Several levels of modularity are achieved. The first level of modularity is at the operational level, and divides the package into smaller programs. These programs correspond to the larger division of the algorithm. For example, the mesh generation and element matrix calculation is programmed as one program. At this level, modularity means not only structural organisation of the package but also allows the re-start facility. As a result, in case of a local error in any program, all the results from the preceding programs can be saved and the operation restarted at the point of error. Within every program, another level of modularity based on logical division is achieved. Every program, therefore, contains one main program and many subprograms performing independent functions. The third level of modularity is not as explicit as the previous two. A subprogram generally consists of a number of blocks of statements, each block corresponding to certain identifiable operation. To manifest this modularity these blocks are generally separated by COMMENT statements.

Statement labels are given in ascending order within each subprogram. For easy reference, FORMAT statements are placed immediately after the corresponding READ or WRITE statements, except in cases where previously defined FORMATS are used. The input formats are of free format type. A systematic and descriptive nomenclature for program variables has been attempted. Such an approach is restricted only by the fact that a FORTRAN variable can have only six characters.

In addition, extensive COMMENT statements are incorporated to describe all the input variables, the dimensioning of the arrays and most of the other variables appearing in the program. Checks on the adequacy of the storage of the dimensioned variables are incorporated wherever possible.

With the increase in sophistication of the program the likelihood of an execution time error increases, and the operating system error diagnostic facilities are generally complex. To forestall the occurrence of an operating system error, which could be expensive if it arises in the later stages of a program, extensive error traps and error diagnostic capabilities are implemented within the programs. Most of these precautions consist of checks on the consistency of the input data, availability of adequate storage and wherever possible the intermediate results. In case an error is detected, the approximate location and the values of some key parameters are printed out, and further running of the program is aborted.

The analysis involves a large number of arithmetic operations and it is therefore important to consider the precision and efficiency. Initially, the programs were developed on an ICL-1903T computer, which has a precision of 11 significant digits in single precision. The precision was found sufficient for the analysis. Later the programs were transferred to a GEC-4070 mini-computer with a virtual memory system, for the production of results. As the single precision of a GEC computer is only about 6 to 7 significant digits, it was found inadequate due to the occurrence of round-off damage. The programs were converted to double precision which caters for about 15 significant digits. It was found that ICL single precision results were nearly as accurate as the GEC double precision results, inferring that 11 significant digit precision was about the requirement. As the double precision programs increase the requirement for core and time, to improve the efficiency, single precision is retained for programs where precision has no significant effect.

A large size finite element analysis usually requires a vast amount of input data. Often it is one of the major difficulties in using such methods of analysis. A suitable mesh generation scheme has been devised to minimise the input data. However, even with such a facility, one would be dealing with a considerable amount of input data which is prone to human error. Such errors may still escape the error trap mechanisms in the program, however extensive they may be, often causing a wastage of valuable computer resources. The errors which are likely to escape the human inspection are normally in the category of incorrect nodal coordinates or wrong nodal description. In either case, the result is a distorted or sometimes badly disfigured structure. The best way of locating such errors is to provide facilities in the program to project the finite element meshes graphically, before the analysis proceeds further. Keeping these factors in mind, graphics programs are developed into the package. These present the finite element mesh and a three-dimensional view in a form convenient for human inspection. The graphics programs make extensive use of the GINO-F library, and the

logic developed is such as to show up the errors as clearly as possible in the graphical illustrations.

Output from a large size finite element analysis, in the form of displacements and stresses, is vast in quantity. Analysing the results in such a form usually makes a heavy demand on human resources. To minimise the human effort and help with a better presentation, an output processing program has been developed. The program employed in this study uses the systems GINOGRAPH and GINO-F libraries and presents the results systematically in the form of graphs.

3.3 PROGRAM LAYOUT

The package is made up of three programs for operational purposes. Each program corresponds to a major phase of the solution, and may be referred to as a Jobstep. With proper Job control commands, the three programs can be run as a single Job for a complete solution. The first, which is basically an input program, generates the finite element mesh using the card input and calculates the element matrices. The second program, which corresponds to the frontal solution, assembles the element matrices and solves the resulting equations for the displacements of the structure. It also evaluates the element stresses and the nodal averages of the stresses. The last program, which is an output processor, organises the output from the frontal solution and presents the final results in the form of graphs and tables. More details of the programs are given in the following section. The overall flow diagram of the package is shown in Fig 3.1.

3.4 INPUT PROGRAM

Logically the program may be divided into three main phases consisting of mesh generation, plotting and the calculation of element matrices. The general flow diagram for the program is shown in Fig 3.2.

The input to a finite element analysis clearly requires as much attention as any of the other important factors. Though the input can be described in simple terms as material properties, nodal coordinates and element nodal descriptions, the quantity of information is vast. The coordinates and the element descriptions require input of thousands of numerical values. The preparation of such a quantity of input information from drawings, not only requires a large human effort but also is susceptible to the almost inevitable human errors. To minimise the input effort, an attempt has been made to devise a mesh generation scheme usable for centrifugal impellers, which is described in the following section.

Mesh Generation Scheme

While it is clearly desirable to devise a mesh generation scheme which could reduce the input to a very few parameters, often there are other considerations which are important and which make such an exercise very difficult. If a mesh generator is devised it should be capable of describing all the possible geometries for the component considered and should not generate element geometries which are awkward from the point of view of analysis. Often these considerations restrict the manoeuvrability in the planning of a mesh generator and make the reduction of input beyond a certain point impracticable. In the case of centrifugal impellers, the geometric variations possible in practice are great. The cross-sections of the disk and the hub can vary, the number of blades can vary and the impeller may have a cover disk or a shroud. In addition, the most important variations are in the blade geometry, since it can have different shapes and can vary in thickness and curvature from point to point in both the radial and axial directions. Keeping these possibilities in mind, a scheme has been devised with which to generate three-dimensional meshes for centrifugal impellers, using 20 and 15 node isoparametric elements.

To describe the scheme, an impeller modelled with a simple coarse mesh is considered. The finite element model of a repeatable segment, which is required for the analysis, is shown in Fig 3.3(a). In the scheme considered, the segment is considered to be made up of layers, as shown separated in Fig 3.3(b). The number of layers will be odd and the mid-layer will be compatible with the outer layers so as to join the blade as shown in the figures. The nodal configuration will be completely described by the mesh represented by the Figs 3.3(c) to 3.3(f), for the above example. In the nomenclature used X-X planes are common surfaces joining two adjacent layers and Y-Y planes are mid-layer surfaces. In the above nomenclature, the word 'plane' has not been used in its strict geometric sense. For a three layer idealisation, as in the case of the current example, there will be four X-X planes and three Y-Y planes as shown in the figures. Recollecting the nodal configurations of the 20 and 15 node elements, as illustrated earlier in Figs 2.2 and 2.3, it is evident that the element cross-sections in X-X planes will have both corner and mid-side nodes, while in Y-Y planes only corner nodes will exist. The nodes in the disk will be numbered first followed by the nodes in the blade as shown in Figs 3.3(c) to 3.3(f) for the given example. The element numbers as marked in these figures are given layer by layer, with the blade following immediately after the disk layer to which it is joined.

The generation of nodal coordinates and the element nodal descriptions as achieved by the scheme are explained using the above example. The nodal coordinates are generated in two steps, first the R-Z coordinates and later the θ coordinates. The R-Z coordinates of the nodes 1 to 39 of the first X-X plane of the disk are input to the program. For the nodes 40 to 55 of the first Y-Y plane of the disk, the correspondence of image nodes in the first X-X plane are input, so as to

assign corresponding R-Z coordinates within the program. With the basic parameters, such as the number of layers, already known, the program generates the node numbers and the R-Z coordinates for all the remaining nodes in the disk. Similarly, the R-Z coordinates of the nodes 205 to 221 of the first X-X plane, followed by the nodal correspondance for the nodes 222 to 227 of the Y-Y plane of the blade are input, to generate the R-Z coordinates for all the nodes in the blade. The program has an interpolation scheme for calculating the coordinates of nodes placed equidistance between any two nodes. Hence, in the above example it is enough if coordinates of the nodes 3 and 7 only are input leaving the nodes 4,5 and 6 to be interpolated. Proceeding further, the θ coordinates of all the nodes in the two X-X planes of the blade, and the disk layer below are required to be input. This was considered necessary so as to account for the variable blade thickness and curvature from point to point in a general case. Since the measurement of θ coordinates from a drawing would be prone to error, the linear dimension $R \cdot \sin(\theta)$ measured from a reference axis (fixed at zero θ) are to be input to the program instead of θ . This linear dimension may be called x. In the case of current example, the x for the nodes 56 to 94 and 111 to 149 of the two X-X planes of the disk are input to the program. Immediately following the above, in order to establish the relationship of other X-X planes relative to the above planes, the x of the first node of each X-X plane prior to the above planes is required as input. In the current example, the x for the node 1 is input to the program. Using the above information, the program generates the θ coordinates for all the nodes in the disk. Similarly, the x values for the nodes 205 to 221 and 228 to 244 are input to the program to generate the θ coordinates of all the nodes in the blade. Once again the interpolation is available for equidistance nodes in the above input. Next, the element nodal descriptions of the elements 1 to 8 in the first layer of the disk and elements 17 to 21 in the blade are input to the program. The program generates the element nodal descriptions for all the remaining elements. A close observation will reveal that the nodal description of element 2 can be generated using those of elements 1 and 3. Similarly, for elements 7 and 20 from the respective adjoining elements. For finer meshes it is possible to interpolate for more elements sandwiched between any two elements. The program has an interpolation scheme to reduce the input in such cases.

Though the example considered is of a radial bladed impeller, it will be noticed that the data input takes no advantage of the radial property of the blade. If the same impeller had a curved blade as shown in Fig 3.3(g), the input would remain identical except for a change in the values of x supplied in order to generate the θ coordinates.

Plotting

The diagrams shown earlier in Figs 3.3(a) to 3.3(g) were drawn with the aid of a graph plotter and were output from the first program. Extensive use of the GINO-F graphics library is made by the program routines which are devised to draw these diagrams. From observation of

the diagrams it can be appreciated that the programming associated with this type of problem will inevitably be complex, and that considerable knowledge of the graphics library is required. A complete description of the graphics routines is considered to be outside the scope of this text as it would require detailing of the routines. Only the broad outlines are described in the following text.

The paper size nominated to the plotter for drawings corresponds to A4 size paper and a 15 mm margin is allocated all around. This choice was made from the point of view of presentation of the diagrams in reports etc. As the plotter units are in millimeters, an input parameter is provided to supply a value for the scaling which can take in to account the input data units and any magnification or contraction required for drawing the diagrams. However, an artifice is provided in the program to make sure that the supplied scale value does not produce diagrams which are too large or too small compared to the size of the paper. If such is not the case then, the program will choose a suitable value for the scale, based on the tip radius of the impeller, so as to make optimum use of the paper.

In the mesh, the nodes are marked using the corresponding R-Z coordinates. Any errors in the R-Z coordinates will show up clearly as displaced nodes in the diagrams. The mesh is developed element by element and the element nodal descriptions are used to draw the cross-sections. Only the nodes describing the face with corner nodes 1 to 4 of 20-node elements (Fig 2.2) and 1 to 3 of 15-node elements (Fig 2.3) are used to draw the R-Z mesh. Any disfigurement of the mesh can be attributed to the errors in the element nodal descriptions. However, a correct R-Z mesh is not a complete check on the element nodal descriptions as only nodes on one face of each element are used for drawing the mesh.

Three-dimensional views are produced to help give further checks on the input data. A complete view is drawn so as to give a good representation of the structure assembled with the elements. In addition, for finer detail the same view is produced with layers of the disk drawn on separate sheets. Though the views are drawn to scale, it may be noted that the actual dimensions in the drawings will also depend on the position of the viewpoint, fixed with respect to the axes of the drawing. The views are developed by drawing element by element using the element nodal descriptions. Two pen colours are used to draw alternate layers of the disk and a third colour for the blade, for an easier inspection of the diagrams drawn. Any disfigurement of the elements indicates errors in the element nodal descriptions. A broad check on the θ coordinates is possible with the help of the above views. However, further checks on the θ coordinates is advisable by means of a close inspection of the nodal coordinates printout. In the views produced, straight lines are used to draw the element sides in place of the actual curves because of the non-availability in the graphics library of suitable curve routines usable with three dimensional drawings.

Element Matrices

For the creation of element matrices, a common main subroutine is adopted for both the 20 and 15 node elements, so as to achieve compactness. Based on the type of element, this subroutine will pick up the appropriate integration points and the shape functions. The shape functions for the two elements are built in to separate subroutines. The element routine also calculates the load vector due to centrifugal forces caused by rotation.

3.5 FRONTAL SOLUTION

This is the second program in the package and the general flow diagram for it is as shown in Fig 3.4. The basic program is adopted from the original paper of Irons (Ref 31). It is extended to incorporate prescribed displacements to any of the variables in the solution and to evaluate the nodal averages of the stresses. All the extensions made are such as to leave the original coding of the program intact, to enable easy reference from the paper. To understand the frontal concept, and the original program, one is advised to refer to the above paper. While making the extensions, every care was taken to keep open all the user options in the program such as complete solution, element by element solution, etc. All the new variables introduced are defined clearly by means of Comment statements. To identify the additions more easily, the new statement numbers are given in the nine hundred range, which was not used earlier in the program. In line with the original program, checks on the adequacy of storage for the new dimensioned variables are incorporated in to the program. To improve the performance of the frontal reduction, the program is amended to begin the elimination of the variables from the end of the front (Refs 44,49). This is the only case where the original code is altered.

To calculate the element stresses suitable element routines are added to the program. Once again to achieve compactness, a common main routine is prepared for calculating the stresses for both the 20 and 15 node elements. Based on the type of element, it will pick up the appropriate sampling points and the shape functions. The shape function routines are identical to those of the first program. As proposed in the analysis, the stresses in an element are calculated at the sampling points and are extrapolated to the element nodes by the method of least squares. To improve the performance of the technique, the stress smoothing matrices, relating the stresses at the element nodes to the sampling points, are calculated only once outside the program and are incorporated in the program as Block Data, for the two elements considered.

The strategy adopted for incorporating prescribed displacements is one of the well established numerical techniques. As soon as a particular variable is mature and is ready for elimination, a very large

number is added to the corresponding diagonal element of the matrix and the product of the large number with the prescribed displacement is added to the corresponding load vector element. To evaluate the nodal averages of stresses, the basic frontal concept is extended and the code generated by the pre-front program is utilised. Statements are added to the program to write the results to an output tape, for use as an input to the next output processor program. An artifice is created to identify the different type of records corresponding to stresses, displacements, etc. As mentioned previously in article 2.9, the computer core requirements for a frontal solution of a given problem can be minimised by a judicious choice of element sequence to the front. In the present problem of centrifugal impellers, with the proposed mesh, it is suggested that a good choice would be to progress from root to tip or vice versa in determining the sequence of elements to the frontal solution. For the present case, this is the last input given in the first program.

The program as developed for the package may be considered independent and easily usable for a different problem. The only changes which may be necessary to adopt will correspond to the element stress routines if different type of elements are used.

3.6 OUTPUT PROCESSOR

The need for a program to process the output results from a finite element solution increases with the size of a problem. Static analysis of a typical impeller would involve a solution of the order of two to three thousand simultaneous equations. The output from the frontal solution program, consisting of displacements, stresses, etc, will involve tens of thousands of numbers. Analysing such results in the form of printed output would inevitably demand considerable human effort. With the computer core space being at a premium, it is often necessary to use the option of element by element solution in the frontal program. The results will be in the form of output for each element, hence increasing the need for an output processor.

Every facility has its accompanying restrictions. To devise an output processor program, it becomes necessary to decide upon a layout for the presentation of the results. This will involve the location of the critical zones of the structure considered, the form in which the results are presented and the format of presentation. It is not always a simple process to choose a layout which is suitable for all situations. In the present case, it was decided to follow the layout developed by Givan and Thurgood (Ref 22). This presents the circumferential and radial stresses at the front and rear of the impellers for the positions beneath and between the blades, in the form of non-dimensional graphs. Reference 22 also presents the non-dimensional radial stresses along the blade leading edge and axial stresses along the bore. In the present program, the above layout is extended to present also the radial and axial displacements of the

impeller in a similar format.

The flow diagram for the program is shown in Fig 3.5 . The program reads the card input identifying the nodes at the front and rear of the impeller at positions beneath and between the blades, etc. It also reads the other input parameters required for the non-dimensionalisation of the stresses. At the next stage, the results from the Frontal solution are decoded from a tape and are organised into arrays. The results are non-dimensionalised as required and are tabulated on the line printer in addition to the plotting of graphs on the plotter.

3.7 EXTENSIONS

The results from the current investigation were mainly planned for the case of centrifugal loading due to rotation, as the experimental results available from previous investigations were limited to that type of loading. The paucity of data available for real temperature and pressure distributions in impellers was another cause which made it difficult to plan a package which is usable for all of these three types of loading. Hence the programs, as described in the preceding sections, were initially developed only for the case of centrifugal loading. However, variants of the programs were later developed to consider the other two types of loading as examples. The changes required, as described in the previous chapter, are simple. The necessary modifications to the programs are listed below.

Thermal Loading

The input in the first program is extended to include the nodal temperatures. In the element routine the load vector is calculated due to the thermal instead of centrifugal loading. Whilst writing the tape, for input to the frontal solution, the element nodal temperatures follow the element coordinates. In the frontal solution, the second program, the element stress routine is modified to subtract the initial thermal strains from the actual strains, before the calculation for stresses proceed. In the third program, the graphs representing the stresses are modified from non-dimensional to dimensional by incorporating suitable scales to the axes.

Fluid Loading

The input in the first program is extended to include the fluid pressure distribution and the identity of element faces which are acted upon by the fluid pressure. The calculation of the load vector is carried out by surface integration of the element faces upon which the fluid is acting, by means of a separate subprogram. The frontal program needs no alteration and in the last program the stress graphs are

dimensionalised with suitable scaling.

Miscellaneous Cases

To study the effect of the number of blades on the stress characteristics, a small subprogram was added to the input program to manipulate and change the input data for an increase or decrease in the number of blades. The effect of the curvature of the blades on the stresses was studied with a similar manipulation, by changing the input data of a radial bladed impeller to that of a curved bladed impeller.

Although it is not an efficient process, with minor changes in the Input program the package was used to analyse axisymmetric disks.

CHAPTER 4

RESULTS

4.1 GENERAL DESCRIPTION

Once it had been established that adequate accuracy could be obtained with a reasonable number of finite elements, a number of impeller configurations were analysed and the results so obtained compared with available experimental photoelastic results from previous investigations. In addition, they were also compared with results from a two-dimensional technique which is generally considered to be the most accurate of such solutions, in order to estimate the improvement which could be expected with a three-dimensional solution. While the above stages were essential if the proposed method of analysis was to be evaluated, additional results were produced to study some important stress characteristics of impellers. For example, the effect of inter-blade bending was studied in more detail by varying the number of blades in a typical model of radial impeller. The effect of curvature of blades, which is hitherto unpublished, was studied by introducing curvature in radial bladed impellers. The calculation of stresses in curved bladed impellers is outside the scope of most of the analytical techniques previously presented. Other important cases such as an impeller with cover plate and an impeller with blades on both sides of the disk were analysed to show the scope of the current method of analysis. All the above results were obtained for the case of centrifugal loading due to rotation alone, since the experimental and theoretical results from previous investigations were limited to that type of loading. However, two further examples were also considered, one with thermal and another with fluid loading, to show the applicability of the present analysis to those cases.

Hawker Siddeley Dynamics Ltd impeller models N, P and R (Ref 22) were chosen as test cases for the current investigation. The geometric dimensions for the models N and P are reproduced in Fig 4.1. The dimensions of model R are identical to model P except that the model R has a flat back throughout.

The results were produced in general for impellers manufactured from steel. However, some results were produced with epoxy as the impeller material so as to enable a comparison with available photoelastic results to be made. The material properties used are as follows.

Steel

Youngs Modulus = 29,000,000 lbs/sq in

Poissons Ratio = 0.3

Mass Density = 0.00074 lb sec²/in⁴

Coefficient of Thermal Expansion = 0.000011 /°C

Epoxy

Youngs Modulus = 500,000 lbs/sq in

Poissons Ratio = 0.495 (as a near approximation to 0.5)

Mass Density = 0.00019151 lb sec²/in⁴

For the case of centrifugal loading, the stresses obtained are presented with a non-dimensional format, similar to that used in reference 22. To non-dimensionalise the stresses, they were divided by the inertia term of a unit volume of the material placed at the tip radius of the impeller and rotating at the same speed as the impeller. The inertia term is given by $\rho \omega^2 r^2 / g$, where ρ/g is the mass density of the material, r is the tip radius of the impeller and ω is the rotational speed of the impeller in radians per second. The impellers were considered to be free at the bore and were spinning freely in the space. The stress constants as given in the non-dimensional graphs are for a speed of 10,000 rpm in the case of steel impellers and 3,000 rpm in the case of epoxy impellers.

Constant stresses $\frac{\rho \omega^2 r^2}{g}$

4.2 ACCURACY AND CONVERGENCE

Whenever a finite element analysis is employed in obtaining a solution for a particular problem it is expected that an exact solution will be forthcoming at the limit of element subdivision. At any stage of a finite subdivision, the solution is only approximate, for example like a Fourier solution with a certain number of terms. What is important to the engineer is to know the order of accuracy achievable in a typical problem with a particular element subdivision. Although it is preferable to assess the accuracy of a specific solution by comparing with an exact solution, it is often not possible to obtain an exact solution for an engineering problem. In such cases, an assessment is made by a study of convergence using two or more stages of subdivision. A useful check in such an assessment would be to study the maximum deviation of nodal stresses, calculated from the elements surrounding a node, with respect to the average of the stresses for the node. Engineering experience suggests approximately 3% as the allowable maximum deviation. With experience, it is possible to assess a priori the order of accuracy which is achievable in a specific problem with a given element subdivision.

To determine a suitable mesh size for the present problem a steel impeller with the dimensions of model P (Fig 4.1) was considered. In the first instance, it was treated with a relatively coarse subdivision, which is shown previously in Figs 3.3(a) to 3.3(f). The stresses and displacements due to centrifugal loading obtained are shown in Figs 4.2(a) and 4.2(b). In this case, the maximum deviation of the nodal stresses was found to be of the order of 10 to 15%, which clearly indicated a need for a further subdivision of the mesh. However, it may be noted that the displacements obtained would be more accurate than the stresses (by an order), since the calculation of the stresses would

involve the differentiation of the displacements (losing an order in accuracy). Proceeding further, the impeller was modelled with a finer mesh as shown in Figs 4.3(a) to 4.3(c), which resulted in the stress and displacement characteristics as shown in the Figs 4.4(a) and 4.4(b). With this subdivision the deviation of the stresses were almost acceptable with a maximum of about 3%, except in the region of 0.3 radius of the disk, where it was of the order of 4 to 5%. This radius represents the transition between the hub and the diaphragm of the disk. Although the results were considered satisfactory from the point of accuracy, the model gave rise to problems when used with high Poisson's ratios nearing 0.5 for the epoxy material. On further study, it became known that 15 node elements are not suitable for such situations. As results were needed with epoxy as material, for comparison with photoelastic results, the impeller was modelled for a third time, but this time using only 20 node elements as shown in Figs 4.5(a) to 4.5(c). Further refinement of mesh was carried in the transition zone of the disk while modelling the above subdivision. Figs 4.6(a) and 4.6(b) show the stresses and displacements obtained using the third mesh, which were found satisfactory from the view point of accuracy.

At this point it is perhaps necessary to draw attention to one aspect of the stresses plotted. An inspection of the front circumferential stresses in Fig 4.6(a) reveals that at about 0.25 radius of the disk there is a steep gradient in the plotted stresses, which may apparently raise some doubts with regards to the accuracy achieved. An explanation for this steepness can be seen in the following example. For the stresses occurring at the location between the blades on the front face, the nodes 1, 10, 15,to 101 (Fig 4.5(b)) are traced to plot the stress characteristic curve. As the nodes 43, 44 and 45 have very similar radii, although they are not closely spaced in the axial direction, any increase in the stresses in that region is reflected as a steep gradient in the curve, as it is plotted against radius. This is very important to note while studying the further results as some times fluctuation in the stresses in that region could result in a fluctuating curve, which may be misleading. In general, it is important to keep the geometry of the impeller in view while studying the results.

4.3 COMPARISON WITH PHOTOELASTIC RESULTS

Once the convergence of the analytical results had indicated a satisfactory mesh size, it was considered essential to compare the calculated results with photoelastic results for a few sample cases of impellers. To assess the improvements achieved by the proposed method of analysis, the results were also compared against results from Schilhansl's finite difference theory (Ref 14). Schilhansl's theory was chosen as it is considered to be the most accurate among the conventional two-dimensional methods of analysis proposed to date.

Five of the Hawker Siddeley Dynamics Ltd impeller models, including two without blades, from reference 22 were considered for the above comparison. The models include impellers with as few as seven blades and as many as seventeen blades. The photoelastic results and the Schilhansl finite difference results as reported in the above reference are adopted for the comparison. Finite element results for the above models were obtained with epoxy as the material and a Poisson's ratio of 0.495 is used as a very near approximation to 0.5. Results were also obtained for a Poisson's value of 0.499, but were found to be very sensitive to the local fluctuations in stresses.

Impeller Model - P

This is a radial impeller with seventeen blades and the finite element subdivision as already shown in Figs 4.5(a) to 4.5(c) is used for the analysis. The non-dimensional stress characteristics as obtained from the analysis are presented in Figs 4.7(a) to 4.7(c). The photoelastic results (thick lines) and Schilhansl finite difference results (broken lines with dots in between) are shown superimposed on the same figures. With as many as seventeen blades, the finite difference results were supposed to give a good approximation to the actual stresses.

Considering the stresses on the rear side of the disk, it will be noticed that the finite element results have shown an improvement over the finite difference results and are closer to the photoelastic results. The accuracy of the results shown can perhaps be appreciated by noticing the precision with which the relatively small inter-blade bending effect is indicated for the case of the circumferential stresses and the proximity of the characteristic curves to the photoelastic results. The inter-blade bending effect is seen as the distance between the stress curves representing the beneath and mid-blade positions. It can be seen that the analytical results deviate quite markedly from the photoelastic results near the bore and hub region. A possible explanation for this is that for the present analysis the impellers were assumed to spin in free space with no restraint whatsoever at the bore or around the hub. This means that the radial stresses at the bore will be zero to satisfy the free boundary condition, resulting in a very high order of circumferential stress. The increase in the circumferential stress near the bore will be almost exponential which is given by the analytical models. This being the theoretical state, in practice a photoelastic model will never achieve such a boundary condition, since a shaft or a fixture is needed to support the model resulting in some constraint at bore. For example a rigid bore would result in a very high radial stress and low circumferential stress at the bore, which is the opposite situation to that for a free bore. The fall in circumferential stress in the hub region for the case of the photoelastic results shown indicates that free bore conditions were not achieved during tests. This may also have resulted in a marginal increase in the radial stresses in the region of hub, which could be the reason for the slightly higher peak for the case of photoelastic

results. In addition, unless vacuum conditons are reached during experimental spinning, there would be certain amount of torque introduced into the impeller at the bore, which could alter the stresses at the bore and in the hub region.

Considering the stresses on the front side, photoelastic stresses were available for only the between the blades position of the impeller. It can be noticed from the figures that while the finite element results are quite close to the photoelastic results (again except in the hub region), the finite difference results are not as good and are a poor approximation, particularly for the case of circumferential stresses. However, this would be expected, since the presence of blades will have a direct effect on the stress characteristics at the front face of the impeller. The finite element results show the presence of the axial stress at the bore, these are assumed to be zero by the conventional two-dimensional theories.

Impeller Model - N

This is a seven bladed radial impeller and the finite element subdivision, as used for the analysis, is shown in Figs 4.8(a) to 4.8(d). The finite element results and the superimposed photoelastic and finite difference results are shown in Figs 4.9(a) to 4.9(c). With as few as seven blades and particularly with a large blade size, considerable inter-blade bending is expected, resulting in a significant difference between the stresses at beneath and between the blades positions of the disk.

Considering the stresses on the rear face of the disk, it is apparant from the figures that the finite difference results are a very poor approximation and may be of little use to the designer, particularly the circumferential stresses. Whereas the results from the proposed finite element method show a good agreement with the photoelastic results. The results also show the capability of the proposed method for adequately dealing with the severe inter-blade bending effect, the analysis remaining the same irrespective of the number of blades present in an impeller. The earlier suggestion that the experimental models did not achieve free bore conditions is again clearly reflected by the high radial stresses close to the bore in the photoelastic results.

Considering the stresses on the the front face of the impeller disk, the improvements achieved by the proposed method are quite significant. While the finite element results are very close to the photoelastic results, the finite difference results can be seen to vary considerably. In Fig 4.9(d), the displacements, particularly in the axial direction, reflect the significant effect of inter-blade bending, due to the relatively few blades in the impeller considered.

Impeller Model - R

This is a radial impeller with seventeen blades and is identical to model P except that it has a flat back. Figs 4.10(a) to 4.10(c) show the finite element subdivision used for the analysis. The mesh for this model was generated from the mesh already developed for model P, by adjusting a few nodes in the disk to represent the flat back. This is a flexibility with finite element modelling, as minor changes in geometry can easily be accommodated with a change of coordinates for a few nodes. The stresses obtained together with the superimposed photoelastic and finite difference results are shown in Figs 4.11(a) to 4.11(c). Most of the comments made for the case of model P are applicable to this case, except that the axial stress in this model is very low at the bore and may apparently satisfy the assumption of zero axial stress used in conventional theories.

Disk from Model - P

Although it is not an economical way to analyse axi-symmetric disks with a three-dimensional solution, the proposed method was used to analyse a few disks, by adopting some minor changes to the mesh generator. The disk of the impeller model P is the same as the unbladed impeller model J referred to in reference 22. The finite element subdivision used is shown in Fig 4.12. Stresses and displacements as obtained are presented in Figs 4.13(a) to 4.13(c). Photoelastic results were not available for this model and hence the results were compared only against finite difference results using the Schilhansl technique. The results in general show a good agreement with each other. However, the presence of significant axial stress in the model, as shown by the finite element results, raises doubts whether the finite difference results can be considered to be completely accurate, even for this unbladed case, as the theory assumes a priori zero axial stress. The axial displacements as shown indicate a significant axial bending of the disk.

Disk from Model - N

This is the same as the unbladed impeller model H referred to in reference 22. Fig 4.14 shows the finite element subdivision used and Figs 4.15(a) and 4.15(b) show the results obtained, along with superimposed photoelastic and Schilhansl results. While there is a good agreement for the case of the circumferential stresses, the agreement is not as good for the case of the radial stresses. However, considerable doubts are raised in the validity of the photoelastic results, as the radial stress is shown compressive both on the front and rear face, near the region of tip radius, which cannot be the case with an impeller under centrifugal loading due to rotation. This cannot simply be attributed to the disk bending, since for such a case the stresses will be tensile on at least one face. As for the case of the previous model,

axial stress is found to be present at the bore of the impeller as shown in Fig 4.15(c). The axial displacements reflecting the disk bending are also shown in Fig 4.15(c).

4.4 Inter-Blade Bending

As pointed out during the review of the literature, the presence of blades in an impeller will result in what is called the inter-blade bending effect, under centrifugal loading due to the rotation of the impeller. With blades on one side of the disk, the circumference of the disk deflects axially in the form of half sine wave between blade root locations. This effect is greatest at the outer radius and disappears towards the inner radius. It is more pronounced for the case of impellers with relatively few blades and results in bending stresses of significant magnitude in the disk, resulting in a variation of the stresses between the blade root and mid-blade locations in the disk. Hitherto, the aspect of inter-blade bending was studied to a limited extent by means of experimental photoelastic tests carried out on sample models of impellers having relatively few blades. They basically served either to highlight or later confirm the presence of the inter-blade bending effect in such impellers. While that helped to explain why the two-dimensional theories would produce grossly inaccurate results for such impellers, no study was conducted to study the magnitude of the inter-blade bending effect with change in the number of blades in an impeller. As the proposed method of analysis has no limitations with regards to the number of blades in an impeller and it takes into account any inter-blade bending present, a study was conducted to investigate the effects on stress and displacement characteristics in an impeller disk with a varying number of blades. The study was conducted to assess the number of blades beyond which the inter-blade bending effect may be marginal, so as to examine the applicability of conventional theories for a reasonable assessment of the stresses.

The Hawker Siddeley Dynamics Ltd impeller model P (Fig 4.1) was selected for the purpose of the above study. Five derivatives of the model were considered, consisting of the original design for 17 blades and four new models with 7, 9, 11 and 13 blades. As mentioned previously, all the models were assumed to be made of steel and to be rotating at 10,000 rpm. Finite element subdivisions for the new models are shown in Figs 4.16(a) and 4.16(b). These subdivisions use the same nodal configuration as given for the 17 bladed model shown earlier in Figs 4.5(b) and 4.5(c). The stress and displacement characteristics for the original 17 bladed model have already been reported in Figs 4.6(a) and 4.6(b). The characteristics for the new four models are presented in Figs 4.17(a) to 4.17(h). A study of the above results indicate the following.

The stress characteristics on the front face of the impeller at the blade root is always different to that at the mid-blade location on the disk, irrespective of the number of blades present on the impeller.

Naturally, this is understandable since the presence of the blades affects the stress characteristics directly on the front face of the impeller. A two-dimensional theory would never be able to represent such a system of stresses accurately, and at best it may be able to indicate the average of such stresses in the case of an impeller with a high number of blades. Once the above fact is accepted, further discussion can be confined to the stress characteristics on the rear face of the impeller, in assessing the magnitude of the inter-blade bending effect with relevance to the number of blades.

The variation in the number of blades appears to have a more significant effect on the circumferential stresses than on the radial stresses. Considering the inter-blade bending effect, which is reflected as the ordinate between the stress curves representing the beneath and between the blades locations on the disk, the following is observed in the case of circumferential stresses. With the lowest point of the blade on the disk being at about 0.26 of the radius of the disk, for the case of the models considered, the inter-blade bending effect is in evidence from only about 0.45 of the radius outwards for the case of the 17 bladed impeller, whereas it penetrates deep towards the bore for the case of the 7 bladed impeller and is felt from as low as 0.15 of the radius. The effect of the inter-blade bending, in the outer half of the radius of the impeller, is to increase the stresses beneath the blade with a corresponding decrease at mid-blade location. The difference between the stresses at the two locations, which reflects the magnitude of the effect, increases as the number of blades decreases and vice versa. In Fig 4.18 the maximum difference is plotted against the number of blades for both the rear and front face of the impeller.

The inter-blade bending effect is also reflected clearly by the axial displacements; which show different displacements for the blade root and mid-blade locations, as shown in the figures. The maximum difference, which occurs at the tip radius, is also plotted against the number of blades and is shown in Fig 4.18. On observing the plots in this figure, it is only possible to say that the inter-blade bending effect is less pronounced after about 13 blades in the case of the impeller model considered. As a large blade and a less stiff disk can result in more inter-blade bending, the above figure of 13 may not be applicable to all impellers as a general rule. However, the above study may give a reasonable guidance in assessing the number of blades beyond which one can hope for a marginal inter-blade bending effect.

The other observations made during the study are as follows. The circumferential stress at the bore and the maximum radial stress decrease marginally as the number of blades in an impeller fall, reflecting the reduction in blade loading on the disk. The axial stress at the bore increases as the number of blades fall, and is greatest for the unbladed case. However, as the axial stress characteristics at the bore appears to differ from impeller to impeller, the above may not be true in all cases.

4.5 IMPELLERS WITH NON-RADIAL BLADES

There has been little or no previous work published on the effect of blade curvature on the stress characteristics of radial impellers. Even the photoelastic experiments reported in the literature, appear to be limited to radial impellers and were basically meant to evaluate the various available two-dimensional theories. As the curvature of the blade is clearly outside the scope of any two-dimensional theory, no effort seems to have been made to study its implications. As the method of stress analysis proposed here has no such limitations, a study was undertaken to investigate the above, and is presented as follows.

Once again the Hawker Siddeley Dynamics Ltd impeller model P was considered for the purpose of the above study. Curvature was introduced into the radial blade by twisting the blade from the radial plane by an angle which is proportional to the radius and varies from zero at bore to a chosen maximum value at tip radius. Four variations of the model were considered as shown by the finite element subdivisions in Figs 4.19(a) and 4.19(b), consisting of two 17 bladed impellers and two 7 bladed impellers. The angles 30 and 45 degrees as indicated in the figures show the maximum angle of inclination of the tip of the blade from the radial plane. It also indicates a measure of curvature of the blade. As the loading is due to the centrifugal force caused by rotation, negative or positive curvature has identical effect on the stress characteristics. The finite element subdivisions also indicate a typical repeatable segment for the case of impellers with curved blades. It may be recalled from the previous chapter that, due to certain limitations of the graphics libraries, the nodes on the element sides are joined by straight lines in the 3-D views produced. However, in the present analysis, the true curved geometries are taken into account by the isoparametric elements used. In Figs 4.20(a) to 4.20(h), the stress and displacement characteristics determined from the analysis of the above models are presented. A study of the results indicates the following.

Considering the circumferential stresses on the rear face, with increase in blade curvature, the stresses decrease marginally in the region of the bore and the hub, say up to 0.3 of the radius of the disk. This may be of some importance for the case of impellers with a large number of blades, in which case the maximum stress which occurs at the bore would decrease. With increase in blade curvature, there is a marginal increase in the inter-blade bending in the region of 0.3 to 0.8 of the radius of the disk, which would be significant for the case of impellers with few blades, as it would increase the maximum stress which lies at about 0.7 of the radius. However, the predominant effect of the curvature seems to be in the region of the tip radius, where it not only cancels out the inter-blade bending effect present but also introduces a reverse effect as the curvature increases. This may not be of much engineering importance as it would not affect the maximum stress occurring in the impeller. Nevertheless, it is preferable to be aware of such sources of stress. On the front face of the impeller the

curvature appears to have little effect except near the tip where its influence is similar to that discussed for the rear face.

Considering the radial stresses on the rear face, the effect of blade curvature is more or less localised near to the tip, where it reverses the effect of inter-blade bending with increase in the curvature of the blade. On the front face, the effect of the curvature is significant, since it helps to reduce the stresses by as much as 15% in the case of a 45 degree tip curvature. However, the beneath the blade stresses appear to register a marginal increase in the upper half radius of the disk, with increase in the curvature.

Considering the other characteristics with increase in blade curvature, there is a marginal increase in the radial stress on the blade leading edge, while there is hardly any effect on the axial stresses at the bore of the impeller. The axial displacements indicate a general increase of the order of 15% for the 17 bladed case and 30% for the 7 bladed case, for 45 degrees curvature, inferring that curved blades are less helpful in reducing disk bending. The axial displacements also clearly reflect the reversing of the inter-blade bending effect near the tip radius of the disk, with increase in curvature. The radial displacements, which are not reported, were found to have a very minor increase with curvature.

4.6 OTHER DISK CONFIGURATIONS

It is often possible to find impellers of different configuration to the conventional bladed disk, for example such as those with a cover disk or shroud, or with blades on both sides of the disk, or a combination of these. Some of these configurations are not easily analysed by the conventional methods of analysis, because of some of the initial assumptions involved in the formulation of those theories. As the method of analysis proposed here has no limitations with regards to such geometries, the treatment of these cases will be the same as for any other typical case considered earlier, except that some engineering commonsense may be required at the data preparation stage. To illustrate the adaptability of the present technique, two further impellers were considered for analysis, one with a cover disk and the other with blades on both sides of the disk as described below.

A cover plate of 0.065 inch thickness was added to the impeller model P (Fig 4.1), so as to obtain a model with a cover disk. The finite element subdivision and the mesh as used is shown in Figs 4.21(a) to 4.21(c). To prepare the data, the cover disk was assumed to be a part of the main disk, as shown in the finite element mesh. Except for this simple assumption, the data preparation and the analysis remained the same as for any other case. A similar assumption is required if an impeller possesses a shroud. The results obtained, for the above impeller with a cover disk, are presented in Figs 4.22(a) and 4.22(b).

A study of the disk stresses indicated apparently no significant changes in the stress characteristics produced by the cover disk, except for a marginal increase in stresses on the front face near the region of the hub. The radial stresses at the blade leading edge, which is now a part of the inner face of the cover disk, showed a marginal increase. There is a significant reduction in axial displacements indicating an overall reduction in the disk bending.

For the second example, the Hawker Siddeley Dynamics Ltd model R, which has a flat back was assumed to represent a symmetric half of a double sided impeller, for convenience. However, the number of blades was changed to 7, from the original 17, so as to examine clearly the absence of the inter-blade bending effect in a double sided impeller. The finite element mesh remained the same as that shown earlier for model R in Figs 4.10(b) and 4.10(c). The analysis also remained the same except that, as a boundary condition to represent the symmetric half, zero axial displacements were prescribed for all the nodes falling on the rear face of the finite element model, which now represents the plane of symmetry for the double sided impeller. The results obtained are presented in Figs 4.23(a) and 4.23(b). The notation 'rear' as used in the figures now represents the symmetric plane of the disk. A study of the results together with a comparison with results for a single-sided 7 bladed impeller (which are not reported), indicated the following. The inter-blade bending which was significant in the case of a single-sided model is totally absent in the double sided impeller, as is only to be expected. Both the radial and circumferential stresses registered a decrease, of an order of about 20 to 25% in the double sided impeller.

4.7 THERMAL LOADING

As detailed previously in Chapter 2, the scope of the proposed method of analysis is not limited to the case of centrifugal loading, which happens to be the case for most of the previously available methods. With minor modifications to the programs, as described in the previous chapter, the analysis is easily extended to treat the other two types of loading possible, i.e. thermal and fluid loadings. The following example was considered to illustrate it for the case of thermal loading.

A steel impeller as of model P (Fig 4.1), was considered with a fictitious temperature distribution of 50 to 175 degrees Centigrade, from bore to tip of the impeller respectively, as shown in case A of Fig 4.24(a). The stress characteristics obtained using the present finite element analysis are presented in Figs 4.24(b) and 4.24(c). To cross-check the accuracy of the results obtained, the programs were re-run with a new temperature distribution as shown in case B of the Fig 4.24(a). The gradient for the case B distribution is identical to that for case A and the temperature distribution of case B was obtained by giving a step increase of 50 degrees Centigrade at all positions to the

distribution of case A. The stresses obtained with the new distribution were identical to those obtained with the first distribution, which confirmed the correctness of the results obtained. The displacements were however different, reflecting the step increase in the temperatures throughout. The radial displacements obtained for the two cases are presented in Fig 4.24(d). The stress results as presented are in dimensional form, as there was no reference with which to non-dimensionalise them. The circumferential stresses on the rear face, change from tensile at the bore to compressive near the tip, which conforms with the general pattern of the stresses expected in a disk with a typical temperature gradient as considered. The presence of blades is reflected in the stresses by means of different values for beneath and between the blade locations. The steep gradient in the thermal distribution near the region of the tip is reflected clearly by the circumferential stresses, showing a similar gradient.

4.8 FLUID PRESSURE LOADING

As for the case of thermal loading, the general analysis was extended to solve an example of an impeller with fluid pressure loading. However, in this case the loading was realistic and not fictitious. The pressure distribution was obtained from reference 63, which was an ongoing thesis project of a Pump design and technology M.Sc student. As this M.Sc project was concerned with the design of a centrifugal impeller, the necessary details of the pressure distribution for the particular impeller were readily available.

The geometric details of the impeller are reproduced in Fig 4.25(a). It is a seven bladed impeller with a cover disk and a very deep blade curvature. The impeller also has a long hub. With the geometry of this impeller being more complex than any of the others considered earlier in the investigation, it also offered an opportunity to prove the capability of the program package to treat such models. The analysis was first carried out for the case of centrifugal loading due to rotation of the impeller, to determine the order of stresses and their characteristics. Then the analysis was extended to the evaluation of the stresses due to the fluid pressure loading as shown in Fig 4.25(b).

The finite element subdivision of the impeller, as used for the analysis, is shown in Fig 4.26(a). For a clearer impression, an exploded view is shown in Fig 4.26(b). The finite element mesh of the above subdivision is shown in Figs 4.26(c) and 4.26(d). To simulate the boundary conditions equivalent to the presence of a drive shaft, but not including the effects of a key-way, the circumferential displacements of all the nodes at the bore and the axial displacements of the nodes only on the front end of the bore were restrained. The results obtained are as follows.

The stress and displacement characteristics of the impeller due to centrifugal loading are shown in Figs 4.27 (a) and 4.27 (b). In the same figures, the characteristics of the impeller due to the combined centrifugal and fluid pressure loading are shown superimposed (using thick lines). The stress and displacement characteristics due to fluid pressure loading alone are shown in Figs 4.27 (c) and 4.27 (d). Considering the characteristic curves due to centrifugal loading alone, it can be seen that the inter-blade bending effect due to the presence of relatively few blades is clearly reflected by the characteristic curves. Similarly, the presence of high curvature in the blade is reflected by the curves, by way of a reversed inter-blade bending effect, near the tip region of the impeller. Considering the characteristic curves due to the fluid pressure loading alone, severe disk bending is reflected by the axial displacements and the radial stresses. It is less severe at the beneath the blade location showing the influence of the blade stiffness. The negative radial displacements clearly reflect the resultant inward load on the blade, owing to the fluid pressure distribution. Considering now the characteristic curves due to the combined centrifugal and fluid pressure loading, it will be noticed that the inter-blade bending effect increases significantly, due to the addition of pressure loading. In effect, the peak stresses show an increase of the order of about 30 to 50%. The characteristic curves for the axial stresses along the bore and the radial stresses along the blade leading edge are shown in Fig 4.27 (e), for both the cases of loading. The pressure loading can be seen to have only a marginal effect on the axial stress characteristics.

To sum up, the radial and circumferential characteristics of an impeller are significantly affected by the addition of fluid pressure loading to the centrifugal loading. As the peak stresses increase significantly for this particular impeller, it may be unwise to consider only the centrifugal loading, when designing centrifugal impellers in general.

CHAPTER 5

AN EXTENSION TO DYNAMIC ANALYSIS

5.1 GENERAL DESCRIPTION

Having achieved a good degree of success in the application of three-dimensional finite element techniques to the static analysis of impellers, it was decided that an attempt should be made to extend the work into dynamic analysis. The dynamic analysis of an impeller is a problem in its own right, and is expected to demand considerable computer resources. In view of the limited period of time and computer resources that were available, the initial intentions were only to make a survey of the various techniques available in general for dynamic analysis and study in particular their practical applicability to the dynamic analysis of impellers.

However, as it worked out later, a fair degree of success was achieved in application of a new technique for the dynamic analysis of impellers. The technique consisted of a Fourier method of analysis coupled with dynamic condensation. Further progress on the work was mainly restricted by the limited capacity of the GEC-4070 mini-computer available for use.

In this chapter, an effort is made to outline the above work, particularly with the intention of forming the basis for any future work on the subject. Owing to the practical limitations such as the time available for writing and the inevitable volume of the thesis, an indepth description of the technical details is not attempted. However, for the benefit of a reader who may wish to go into such details, a complete list of references is provided.

5.2 ALGORITHMS FOR DYNAMIC ANALYSIS

The characteristic eigenvalue problem for the dynamic analysis of a structure takes the general form (Ref 30),

$$[K][\phi] = \omega^2 [M][\phi] \quad \dots \quad (5.1)$$

where $[K]$ is the stiffness matrix
 $[M]$ is the mass matrix
 ω^2 is a diagonal matrix with eigenvalues
 and $[\phi]$ is the matrix of associated eigenvectors

Historically, this problem has attracted considerable interest, and as a result, a variety of eigensolution techniques are available in the literature. Although, theoretically speaking, almost all algorithms are equally capable of solving any eigenproblem completely, their universal suitability is severely restricted by their differing demands on available resources. Moreover, a complete eigensolution is rarely attempted except in small problems (say fewer than 100 variables). In most practical cases, only a partial solution of the eigenspectrum would

be sufficient, which is duly reflected in the algorithms available. Furthermore, some of the eigensolvers devised are meant to suit some particular special characteristics of the operator matrices. For these reasons, the choice of algorithm for a given problem is severely restricted. In the following, a brief summary of the algorithms available and their practical applicability is described (Ref 50).

The Generalized Jacobi and the Householder-QR-Inverse Iteration methods appear as a first group. These are most efficient when a complete eigenspectrum is required. These methods are not suitable for large problems since they involve continuous transformation of the original matrices, requiring their presence in the computer core all the time. In the next group appear the Sturm Sequence and the Determinant Search methods, which are suitable for large systems with small bandwidths. Since they are most efficient as incore solutions, they become prohibitive for systems with large bandwidths or for very large systems. For large systems with large bandwidths, when only a few of the lowest eigenvalues and eigenvectors are required, the Subspace Iteration method (Ref 51) is most suitable. Each iteration in the method involves a solution of a static problem, equivalent to the full size of the problem considered, plus an eigensolution of a reduced system of an order equal to the number of eigenvalues and eigenvectors sought. In essence, the Subspace Iteration method is equivalent to solving a series of static problems by virtue of operation. This method is most efficiently adopted for use with the frontal concept, which provides for resolution facility as required for the iterations (Ref 44).

In a different category comes the Dynamic Condensation (Refs 52,53,54), which is strictly speaking not an eigensolver, but is primarily used to reduce the size of a large problem. In this algorithm, the larger eigenproblem is condensed to a smaller eigenproblem in terms of some key variables. The resulting system is then solved using any of the above methods. This method gives only an approximate solution at the lower end of the spectrum and it may be attempted only when such a solution is desired. Once again, this method is more easily adoptable when the frontal concept is employed (Ref 44). The selection of the key (master) variables may be automatised by use of the procedures given in references 55 and 56.

In either the Subspace Iteration or Dynamic Condensation methods, an eigensystem with dense coefficient matrices has to be solved as an incore problem. The size of the matrices will generally be small and a complete solution will be required to a high degree of accuracy. For this purpose the Generalised Jacobi method is most suitable.

For more details on the above eigen solvers the reader may see references 45, 50, 57 and 58 are recommended.

5.3 CHOICE OF A SOLUTION

It is essential to consider the various aspects such as the size of the basic problem, the band-width of the system matrices, the possibility of a reduction in the size of the basic problem etc, before an assessment is made of the suitability of any of the algorithms described in the previous section. The first and most important factor which directly affects the dynamic analysis of an impeller is that the principle of repeatability, as used in the static analysis, is no longer applicable, since an impeller can have both symmetric and non-symmetric modes of vibration. In effect, the whole impeller must be considered for dynamic analysis, unlike a single repeatable segment in the case of static analysis. Fortunately, there is the consolation that, reasonably good eigenvalues can be determined with fewer degrees of freedom than is required for a normal static solution (Ref 30). Taking these factors into account, it is possible to consider the size of a typical impeller problem. If the Hawker Siddeley Dynamics Ltd impeller model P (Fig 4.1) is considered, the finite element mesh as shown in Figs 3.3(a) to 3.3(f), which was considered coarse for the static analysis, may be considered adequate for a dynamic analysis. The number of variables (degrees of freedom) in a single segment with such an idealisation is of the order of 732. For a full impeller with ten or more blades, the size of the problem would be of the order of at least seven thousand variables with a band-width of about one thousand. The number of frequencies which may be of interest would be at most of the order of a hundred corresponding to the lower end of the eigenspectrum. If the practical suitability of the algorithms described is considered, for the problem considered, it is apparent that the choice is severely restricted. In essence only Subspace Iteration is feasible and even that algorithm would produce a problem equivalent to ten times or more the size of the corresponding static problem. Added to this there would need to be a number of resolutions of such a problem in order that the necessary iterations could be carried out. Without doubt, such an analysis would require a fast computer with large core and peripheral processing facility. In view of the limited available computer resources and the very high cost of such an analysis, the possibility of proceeding with such an attempt did not arise.

As an alternative approach, it was considered useful to investigate the possibility of carrying out an approximate analysis, which could reduce the size of the problem to that of a single segment. A survey of the literature indicated no such information on the subject, except a brief note (Ref 59) which was found to be of little use. Later, Bennett (Ref 60) suggested a Fourier technique for an approximate solution, which can reduce the size of the problem to a single repeatable segment (ie geometric repeatability). The Fourier method as suggested is separately described in Appendix-A. As envisaged in the method the problem would reduce to a complex eigen system of an order corresponding to the size of a single segment, but solved in turn for a number of harmonics (ie diametral modes). The complex matrix as obtained for a final eigensolution will be of general form with no special features, to consider any further savings in the eigensolution. If this method is

adopted for the analysis, it would be necessary to have a computer capability with which it is possible to solve such a large eigensystem. Hence, even this technique as it stands, would require resources which are not available. However, it was proposed to try a method which would involve further approximation, but would at least enable a further reduction in the size of the resulting eigenproblem to be achieved. This method envisages the use of Dynamic Condensation prior to the application of a Fourier solution suggested by Bennet. The method as proposed is described below.

5.4 PROPOSED METHOD

In the method proposed, a single repeatable segment of an impeller is considered for the purpose of dynamic analysis. As a first step, the system matrices $[K]$ and $[M]$ (stiffness and mass matrices) are condensed to a smaller size, by using the method of Dynamic Condensation (Refs 52,53). The condensed matrices represent only a smaller number of key variables. However, it should be noted, that the effect of the condensed variables is approximated and not neglected. Moreover, their values can eventually be obtained through the key variables, though with a reduced accuracy. Following the condensation, it is proposed that the Fourier solution as described in Appendix-A is now performed on the smaller condensed system matrices. The resulting complex eigensystem is solved in turn for each harmonic (diametral mode) and the eigenvalues and eigenvectors are obtained in the complex form. Using the complex eigenvectors, the eigenvectors in real form may be obtained for any or all of the segments of the impeller. The eigenvectors as obtained represent only the key variables and therefore are expanded, to represent all the variables, by using the method of back-substitution, for the dynamic condensation carried out earlier. The expanded eigenvectors represent the mode shapes of a segment, though only approximately.

Implementation of the procedure as described above would require a much smaller computer facility, than would any of the procedures discussed earlier. However, it is essential to undertake extensive testing of the procedure to establish its viability. The computer implementation of the above procedure, as carried out for the purpose of the current investigation, is described briefly in the following.

5.5 COMPUTER IMPLEMENTATION

The computer implementation of the proposed method of analysis was treated as a crash programme, due to the limited period of time that was available for the purpose. In view of that, maximum use of the available programs for static analysis was made, wherever possible, by adopting suitable modifications to the programs. As the programs thus obtained were not purpose built, some of the programming steps may be found to be inefficient. The overall flow diagram for the program

package, for the proposed method of dynamic analysis, is as shown in Fig 5.1.

The first program in the package is concerned with the generation of a finite element mesh and the calculation of element matrices, for a repeatable segment of an impeller. It was derived by making a few simple modifications to the first program (Input Program) of the static analysis package. While there were no changes to the mesh generator, the main modification was concerned with the simultaneous creation of an element mass matrix along with the stiffness matrix, in place of the loadvector. This required only the substitution of a few statements in the main element routine. The statements corresponding to the application of repeatability to the boundary conditions were deleted. With the deletion of the repeatability conditions, the elements to the front may now be sequenced as they appear in the mesh generated, without any increase in the core required for a frontal operation. Accordingly, the statements corresponding to the element order to front were deleted. The last modification was concerned with the writing of the mass matrix to the output tape in place of the loadvector.

The matrix expression used for the creation of element matrices, in the above, is given by,

$$[M]_{60 \times 60} = \int_{vol} [N]_{60 \times 3}^T \rho [N]_{3 \times 60} \quad \dots \quad (5.2)$$

where $[M]$ is the element mass matrix
 $[N]$ are the element shape functions
 and ρ is the mass density of the material

The above expression is similar but simpler than that given in equation (2.11), for the stiffness matrix.

The second program is concerned with the assembly of the element matrices and the application of Dynamic Condensation, by the use of the frontal concept. It was derived by adopting a major part of the second program (Frontal Solution) of the static analysis package, with suitable modifications. The first modification was the addition of an artifice in the pre-front program, in order to read a list of key variables (masters) from card-input. The key variables are read into the program, immediately after the lists of variables for all the elements are read from the element tape, created by the first program. The program, as modified, treats the list of key variables as if they belong to an element appearing in the last. The resulting effect on the code generated by the pre-program, for frontal processing, is to assign longevities to those key variables such that they are retained in the core at the end of the reduction of all other non-key variables. The second modification was the addition of statements for simultaneous assembly and reduction of the mass matrices along with the corresponding stiffness matrices. The new vectors created to treat the mass matrix operations are similar to those employed for the stiffness matrix. The

assembly process produced is common for both the stiffness and the mass matrices. The modification also included the deletion of statements concerned with the operations on the load-vector used for the case of static analysis. Dynamic Condensation was incorporated in the program by adopting the modifications suggested by Abbas (Refs 44,45), to the static reduction. The stiffness coefficients of the reduced equations corresponding to the condensed variables are now held in a permanent tape file, instead of a temporary file as used for the case of the static solution. These equations will be required for the back-substitution process, employed later in the technique, for the expansion of eigenvectors. At the end of the condensation, the condensed matrices may contain some zero columns and rows owing to the frontal destinations employed. Hence, statements are added to compact the matrices, by eliminating the zero rows and columns corresponding to the null variables. The compacted system matrices are written to a tape file for use in the subsequent program. The remaining part of the static program, concerned with back-substitution and calculation of stresses was deleted.

It may be worth pointing out at this stage that the list of key variables chosen must be such that the internal and the boundary variables, for the impeller segment considered, are adequately represented. A complete, or near complete, condensation of boundary or internal variables must be avoided, so as to ensure that the method proposed is technically viable. It is essential that the key variables chosen on both the boundaries must correspond to each other.

The third program in the package, as shown in the overall layout (Fig 5.1), is concerned with the Fourier method of solution as proposed in the analysis. This particular program was built specifically to perform the Fourier solution described in Appendix-A. Its operations may be described briefly as follows. Firstly, the program reads the condensed system matrices from a tape file created by the preceding program. Next, complex system matrices are assembled using the coefficients of the above matrices. Following that, the inverse of the complex mass matrix (which is Hermitian) is obtained and is post-multiplied with the complex stiffness matrix. It may be recalled, that the product matrix $[M'K]$ as obtained, will be of general complex form (with no special features), for the purpose of eigensolution. The program uses suitable NAG library routines for the above inversion and the eigensolution. Once the complex eigenvectors are obtained, the real eigenvectors for any or all of the segments of the impeller may be obtained by a simple substitution. This involves the substitution of real and imaginary components of the complex eigenvector, for cosine and sine amplitudes respectively, in the basic Fourier relations adopted initially in the analysis. For the example considered in this thesis, the real eigenvectors were obtained only for the first segment of an impeller. The real eigenvectors are written to a tape file, for expansion (by back-substitution) in the following program. The complex eigenvalues obtained as above would contain mainly a real part with near zero imaginary part. The number of frequencies for which the eigenvectors are output for each harmonic (in the lower end of the

spectrum) is pre-decided in the program (say 10). The whole solution, starting from the assembly of complex matrices, is repeated for each harmonic in turn. The number of harmonics for which the solution is required depends on the number of segments in the impeller considered.

The fourth program in the package, which is a short one, is concerned with the expansion of the above eigenvectors, by using the method of frontal back-substitution. The eigenvectors as obtained by the previous program are defined only in terms of few key variables. Often, it is impossible to visualise the mode shapes of a structure with such scarce information. Hence, the approximate displacements for the condensed non-key variables are obtained, by using the method of frontal back-substitution, corresponding to the mode shape defined by the key variables. The program was derived by adopting the back-substitution part of the second program (Frontal Solution) of the static analysis package, with suitable modifications. The operations of the program may be briefly described as follows. Firstly, the eigenvectors defined in terms of key variables are taken as an input, one at a time, from the tape created by the preceding program. The back-substitution equations corresponding to the condensed variables are read from another tape file in the reverse order (ie the equation corresponding to the variable condensed last is taken first). A condensed variable is evaluated by substituting the values for key variables and the variables evaluated preceding to it in the corresponding back-substitution equation. The tape file referred to was created, by the second program, during the process of the Dynamic Condensation. At the end of back-substitution, each expanded eigenvector is written to a tape file, for processing by the subsequent program.

The fifth and last program in the package is concerned with the plotting of mode shapes in three-dimensional form. The expanded eigenvectors, as obtained from the preceding program, contain the relative displacements for all the structure nodes, for a given mode shape. However, it would be impossible to physically scan such a vast amount of information and clearly visualise the mode shape. This program is meant to solve such a practical difficulty. It was derived by readopting a major part of the first program, related to the mesh generator and the plotting of a 3-D view, with suitable modifications. The operations of the program may be described as follows. Initially, it reads the geometric data of the impeller considered (same data as that used in the first program) and creates the necessary information related to the mesh, for use with the plotting of a 3-D view. Next, it reads the expanded eigenvectors, one at a time, from the tape file created by the preceding program. Finally, a 3-D view of the impeller segment (in static condition) is plotted using a particular colour pen on the plotter. The view is superimposed upon another 3-D view representing the displaced mode shape, using another colour pen and broken line mode. To obtain a distinct view of the mode shape, the displacements as defined by the eigenvectors are suitably magnified before the above plotting. The process of plotting is repeated for each eigenvector input from the tape.

5.6 RESULTS

As a first step, a classical problem of a ring with free vibrations was considered, to assess the quality of the results obtainable by use of the Fourier method suggested by Bennett (Appendix-A) and to study the implications of the addition of Dynamic Condensation to it, as proposed. A repeatable sector of a ring, idealised with two 20-node brick elements, as shown in the Fig 5.2 was considered for the analysis. The material of the ring was considered to be steel. Initially, the analysis was performed without adopting any Dynamic Condensation, ie with all variables available for the Fourier solution. The lowest two frequencies for each harmonic (diametral mode) as obtained are presented in Table 5.1, along with the corresponding frequencies obtained using classical solutions. The classical solutions for out-of-plane and in-plane vibrations of a ring were obtained from references 61 and 62 respectively. The first two frequencies in the case of zero and first harmonics, which correspond to the rigid body modes (null energy modes) with zero frequencies, are not shown in the table. The frequencies as obtained using the Fourier solution compare very well with the classical solutions, confirming the capability of the Fourier solution adopted for the analysis. As a next step, the analysis was repeated several times, but each time with an increased number of variables being condensed, to study the implications of Dynamic Condensation on the results. The frequencies as obtained are listed in Table 5.2. In choosing the key variables, for convenience, either all the three degrees of freedom at a node were chosen, or alternatively all the three were omitted for condensation. Alternatively, key nodes were selected. A study of the results indicate the following. A drastic condensation of the internal nodes, from 16 to 2, did not have any significant effect on the frequencies obtained. Whereas a condensation of boundary variables showed a significant effect. With the number of nodes on each boundary condensed from 8 to 4, the frequencies showed only a marginal decrease of the order of about 1%. However, further condensation to 2 nodes resulted in either the out-of-plane or the inplane modes being missed, which depended on the particular key nodes chosen. The print output showed that the missing modes were replaced by spurious ones, with frequencies very near to zero.

To sum up, the proposed method of analysis, of adopting Dynamic Condensation prior to the application of the Fourier solution, is in principle viable. However, a drastic condensation of boundary variables should be avoided, whereas it may be permissible to severely condense the internal variables. Drastic condensation of the boundary variables may result in the elimination of some modes altogether, possibly even those which lie in the lower end of the spectrum and having considerable importance.

Table 5.1

Ring frequencies in cycles/sec

Harmonic No	Out-of-plane		In-plane	
	Ref 61	Fourier	Ref 62	Fourier
2	100.295	100.5	114.647	117.82
3	283.678	284.2	338.419	342.8
4	543.928	544.4	661.139	663.9
5	879.648	880.4	1079.426	1079.4

Table 5.2

Effect of Dynamic Condensation - Ring frequencies in cycles/sec

No	Key Variables Node Nos (See Fig 5.2)	Out-of-plane			In-plane		
		Harmonic No			Harmonic No		
		2	3	4	2	3	4
1	All 1-32	100.5	284.2	544.4	117.82	342.8	663.9
2	1-8,13-20,25-32	100.5	284.2	544.4	117.82	342.8	664.0
3	1-8,14,19,25-32	100.5	284.2	544.39	117.82	342.8	664.0
4	2,4,5,7,14,19,26,28,29,31	99.1	280.2	537.3	116.8	339.49	658.1
5	2,7,14,19,26,31	99.1	280.2	537.3	Miss	Miss	Miss
6	4,5,14,19,28,29	Miss	Miss	Miss	115.87	338.0	656.0

EXAMPLE OF AN IMPELLER

Following the above useful experience, it was decided to undertake the dynamic analysis of an impeller as a sample case. A steel impeller with dimensions as of the Hawker Siddeley Dynamics Ltd impeller model P (Fig 4.1), but with only eight blades, was considered. The finite element mesh as shown earlier in Figs 3.3(c) to 3.3(f) was utilised. Owing to the limitation of the maximum core which was available on the GEC-4070 computer used, it was possible to solve only a complex eigensystem of relatively small size. In view of this, only 102 key variables (belonging to 34 key nodes) were considered for the analysis. Twelve key nodes were chosen on each boundary, while only ten key nodes were chosen in the rest of the internal part of the segment. An attempt was made however to rationalise the selection of the key nodes by using the criterion of high mass to stiffness ratio (Ref 55,56), but was found not to be feasible. Hence a selection was made based on an engineering judgement. The key nodes selected were 95, 98, 100, 104, 105, 108, 109, 223, 225 and 227 to represent the internal part of the segment and 2, 6, 14, 17, 22, 24, 27, 29, 32, 34, 37 and 39 to represent the first boundary, along with exactly corresponding nodes on the second boundary. With the above choice, a complex eigensystem of the order 66 was solved to obtain a solution, which is analysed in the following.

Figs 5.3(a) to 5.3(m) show some of the vibration modes as obtained for the impeller considered. The harmonic number and the frequency, to which a mode shape belongs, are marked in the diagrams. The frequencies as shown belong to the lower end of the eigenspectrum, as obtained from the solution. The accuracy of the results can only be verified either by an experimental investigation or at least by solving the problem a number of times with an increase in the number of key variables so as to establish the convergence. An exercise to establish the convergence was not feasible with the available computer resources. Hence, only a qualitative assessment of the results was possible, which is presented in the following.

An impeller vibrating in free space can have four rigid body modes. In an harmonic analysis, two modes each must be represented by the zero and first harmonics. In the above solution, they were clearly demonstrated. Fig 5.3(a) shows the two translatory rigid body modes obtained with the zero harmonic. The associated null frequencies were clearly marked. The other two rotational rigid body modes possible were represented by the first harmonic as shown in Fig 5.3(d). However, small numerical values for the associated frequencies were obtained, as marked in the diagrams, indicating the numerical error involved in the above analysis. The umbrella mode possible with a zero harmonic is represented by the first diagram in Fig 5.3(b). In the same figure, the second diagram represents a mode related to blade excitation. Similarly, all other interesting possibilities of mode shapes in an impeller are represented by the rest of the diagrams in the above figures. To sum up, while the analysis is qualitatively acceptable, it is advised that a study for convergence is undertaken to establish the numerical accuracy.

CHAPTER 6

CONCLUSIONS AND RECOMMENDATIONS

6.1 STATIC ANALYSIS

The research programme has resulted in the development of a viable three-dimensional method of stress analysis, applicable to all types of centrifugal impellers, giving results that are very close to those which experiment has shown to actually exist. The analysis has no known limitations with regards to geometric factors such as asymmetry of disk, blade curvature, presence of a cover-disk or shroud, single or double sided impeller etc. Similarly, it can treat all of the three types of loading possible in an impeller, viz centrifugal, thermal and fluid. The investigation has resulted in the development of a computer program package which is usable for any of the above cases.

The investigation has proved that, by using a correct program layout and data management scheme, it is practicable to adopt a three-dimensional finite element analysis for impellers on a relatively small computer, for example a GEC-4070 computer. It has also shown that the data preparation effort for such an analysis can be reduced to quite manageable limits, by use of a mesh generator. Earlier fears with regards to the practical usability of a three-dimensional finite element analysis for impellers, considering the cost and computer layout, are discovered to be unfounded since it would be well within the means of a small pump company, as shown by the experience of the current investigation.

The present investigation has helped to study, more closely than ever before, the problem of inter-blade bending in an impeller, particularly with regard to the number of blades. It has also provided for the first time an opportunity to study the effect of blade curvature on the stress characteristics of an impeller.

The earlier proposition that the stresses due to fluid loading may be neglected compared to those due to centrifugal loading is found to be incorrect, at least for the test case considered. In fact, the peak stresses were found to increase by as much as 30 to 50 percent, due to the addition of fluid loading.

6.2 DYNAMIC ANALYSIS

The extension of the investigation to dynamic analysis was quite a satisfying experience, particularly considering the limited period of time that was available for the work. A study of the sample results obtained indicate that the method used was a good step in the right direction. However, it would be unwise to lend any further weight to the adequacy of the technique employed until a thorough test on the convergence of the results obtained.

6.3 FURTHER RESEARCH

The quality and accuracy of the results obtained for the case of static analysis is considered adequate and hence no further research is suggested. At best, one may find it useful to adopt necessary operational modifications to the program package to suit individual requirements.

With regards to the dynamic analysis, it would be essential to carry out a test for convergence of the results obtainable by the method proposed, using a bigger computer. In the event of the results being unsatisfactory, one recourse may be to model the whole impeller and use Subspace Iteration, which would inevitably demand a very large computer layout. Possibly a last but useful step would be to conduct an experimental investigation to establish the accuracy of the theoretical results.

REFERENCES

1. J C Maxwell, 'On equilibrium of elastic solids', Paper read to the Royal Society of Edinburgh, 1850.
2. M Donath, 'Die Berechnung rotierender Scheiben und Ringe (The calculation of rotating disks and rings)', Berlin 1912.
3. H Haerle, 'The strength of rotating disks', Engineering, 106, 131-134, London 1918.
4. J L Meriam, 'Centrifugal stresses in impellers', Aircraft Engineering, 15, 257, 1943.
5. S Tumarkin, 'Methods of stress calculation in rotating disks', NACA Technical Memorandum No 1064, Sept 1944.
6. A M G Moody and M L Land, 'Sum and difference method for calculating stresses in disks', Product Engineer, October 1946.
7. S S Manson, 'Determination of elastic stresses in gas turbine disks', NACA Report No 871, 1947.
8. W G Schmittner, 'Determination of centrifugal stresses in a turbosupercharger impeller by means of a stress coat', Presented at the annual meeting of Society of Experimental Stress Analysis, New York, 1950.
9. F Jabureck, 'Die Festigkeit von radial beschaufelten laufadern (The strength of radially vaned impellers)', Ost Ing Archiv, 7, 214-222, 1953.
10. J W Glessner, 'A method for analysing the stresses in centrifugal impellers', Presented at the annual meeting of The American Society of Mechanical Engineers, New York, 1954.
11. R L Brooking, J Brown, and Miss B R Atkins, 'A direct easily computable method for the evaluation of centrifugal and thermal stresses in disks, and stress changes due to plastic flow and creep', NGTE Report R122, July 1956.
12. P G Hodge Jr and J Papa, 'Rotating disks with no plane of symmetry', Journal of Franklin Institute, 263, 505-522, June 1957.
13. A S Kobayashi and P R Trumpler, 'Elastic stresses in a rotating disk of general profile', Int J Mech Sci, 2, 13-29, 1960.
14. M J Schilhansl, 'Stress analysis of a radial flow rotor', Transactions of ASME, Journal of Engineering for Power, 84, 124-130, January 1962.

15. N S Swansson, 'The stress analysis of radial flow impeller', Australian Defence Scientific Service, Aeronautical Research Laboratories, Mechanical Engineering Note 259, August 1963.
16. F P J Rimrott and D Bell, 'Stress and strain determination in impellers of asymmetrical profile', Canadian Aeronautics and Space Journal, March 1963.
17. 'Investigation of the general stress distribution in impeller wheels', Wright-Patterson Air-Force Base, Technical Documentary Report No ASD-TDR-62-1013, Ohio, March 1963.
18. D A Thurgood and Miss V E F Givan, 'Stress analysis of the asymmetric profile rotating disk', Hawker Siddeley Dynamics Ltd, Stress Office Report No 619 (Parts I and II), January 1967.
19. Miss V E F Givan, 'Impeller deflection test on a type H cold air unit', Hawker Siddley Dynamics Report V R 1081, March 1966.
20. I Williams sand W J Harries, 'Photoelastic investigation on the cold air unit fan of the Comet 4', De Havilland Aircraft Company Report PM 9998/61, December 1961.
21. D A Thurgood, 'Stresses in asymmetric disks', Journal of strain analysis, 4, 65-73, 1969.
22. Miss V E F Givan and D A Thurgood, 'Comparison of stress analysis methods for radial flow rotors', Hawker Siddeley Dynamics Limited, Stress Office Report No 630, January 1969.
23. A N Smith, 'Centrifugal stress analysis methods for radial flow rotors', NGTE Note No NT703, June 1968.
24. A S L Chan and R K Henrywood, 'The analysis of radial flow impellers by the matrix finite element method', The Aeronautical Journal of the Royal Aeronautical Society, 75, 850-860, December 1971.
25. R G Patton, 'Stresses in centrifugal fan impellers', Proceedings of Institution of Mechanical Engineers, 187, 309-315, 1973.
26. H Kikuchi, 'Stresses in radial impellers', Ph D thesis, Cranfield Institute of Technology, 1973.
27. I A Biger, 'Priblizhennyi raschet na prochnost rabochikh koles tsentrobezhyf turbomashins dnoikh storonnim vkhodom (Approximate stress analysis for double sided impeller)', Prochnosti i Dinamika Aviatsionnykh Divgatelei, Sbornuk Statei, 1, 1964.

28. N I Bezukhov, 'Ortotropno-Konstruktivnye diski turbomashin s radialnymi lopatami (Structural orthotropic turbine disk with radial blades)', Raschet na Prochnosti, Ustoichivost i Kolebanniya, 1965.
29. O C Zienkiewicz and F C Scott, 'On the principle of repeatability and its application in analysis of turbine and pump impeller', Int J Num Meth Eng, 4, 445-448, 1972.
30. O C Zienkiewicz, 'The finite element method', McGraw-Hill Book Co (UK) Ltd, 1977.
31. Bruce M Irons, 'A frontal solution program for finite element analysis', Int J Num Meth Eng, 2, 5-32, 1970.
32. J G Ergatoudis, 'Isoparametric elements in two and three dimensional analysis', Ph D thesis, University of Wales, Swansea, 1968.
33. J G Ergatoudis, B M Irons and O C Zienkiewicz, 'Curved isoparametric quadrilateral elements for finite element analysis', Int J Solids Structures, 4, 31-42, 1968.
34. S Ahmed, 'Curved finite elements in the analysis of solid, shell and plate structures', Ph D thesis, University of Wales, Swansea, 1969.
35. S Ahmed, B M Irons and O C Zienkiewicz, 'Analysis of thick and thin shell structures by curved finite elements', Int J Num Meth Eng, 2, 419-451, 1970.
36. Karan S Surana, 'Transition finite elements for three-dimensional stress analysis', Int J Num Meth Eng, 15, 991-1020, 1980.
37. A K Gupta and Bijan Mohraz, 'A method of computing numerically integrated stiffness matrices', Int J Num Meth Eng, 5, 83-89, 1972.
38. G R Cowper, 'Gaussian Quadrature formulas for triangles', Int J Num Meth Eng, 7, 405-408, 1973.
39. K Sham Sunder, R A Cookson and J W Rait, 'Integration points for triangles and tetrahedrons obtained from the Gaussian Quadrature points for a line', Communicated to Int J Num Meth Eng, April 1980.
40. Bruce M Irons, 'Round off criteria in direct stiffness solutions', AIAA Journal, 6, 1308-1312, 1968.
41. Bruce M Irons, 'Quadrature rules for brick based finite elements', Int J Num Meth Eng, 3, 293-294, 1971.

42. T K Hellen, 'Effective quadrature rules for quadratic solid isoparametric finite elements', Int J Num Meth Eng, 4, 597-599, 1972.
43. Bruce Irons and Sohrab Ahmad, 'Techniques of finite elements', Ellis Horwood Limited, UK, 1980.
44. Syed F Abbas, 'Some novel applications of the frontal concept', Int J Num Meth Eng, 15, 519-536, 1980.
45. Syed F Abbas, 'Dynamic analysis of shallow shells using finite element mixed models', Ph D thesis, University of Toronto, Toronto, Canada, 1975.
46. E Hinton and J Campbell, 'Local and global smoothening of discontinuous finite element functions using a least square method', Int J Num Meth Eng, 8, 461-80, 1974.
47. J Barlow, 'Optimal stress locations in finite element models', Int J Num Meth Eng, 10, 243-51, 1976.
48. E Hinton, F C Scott and R E Ricketts, 'Local least squares stress smoothening for parabolic isoparametric elements', Int J Num Meth Eng, 9, 235-39, 1975.
49. K Sham Sunder and J W Rait, 'Comments on Some novel applications of the frontal concept', Int J Num Meth Eng, 16, 477, 1981.
50. K J Bathe and E L Wilson, 'Solution methods for eigenvalue problems in structural mechanics', Int J Num Meth Eng, 6, 213-226, 1973.
51. A Jennings and D R L Orr, 'Application of the simultaneous iteration method to undamped vibration problems', Int J Num Meth Eng, 3, 13-24, 1971.
52. R J Guyan, 'Reduction of stiffness and mass matrices', AIAA Journal, 3, 380, 1965.
53. G C Wright and G A Miles, 'An economical method for determining the smallest eigenvalues of large linear systems', Int J Num Meth Eng, 3, 25-33, 1971.
54. B M Irons, 'Structural eigenvalue problems: Elimination of unwanted variables', AIAA Journal, 3, 961-962, 1965.
55. J N Ramsden and J R Stroker, 'Mass condensation: A semi-automatic method for reducing the size of vibration problems', Int J Num Meth Eng, 1, 333-349, 1969.
56. R D Henshell and J H Ong, 'Automatic masters for eigenvalue economisation', Earthquake Struct Dynam, 3, 375-383, 1975.

57. K J Bathe and E L Wilson, 'Large eigenvalue problem in dynamic analysis' ASCE Jnl of the Eng Mech Div, 98, EM6, 1471-1485, 1972.
58. K J Bathe and E L Wilson, 'Numerical methods in finite element analyses', Prentice-Hall, 1976.
59. F W Williams, 'Natural frequencies of repetitive structures', Quart Jnl Mech Appl Math, 24, 285-310, 1971.
60. J C Bennet - A personal communication through R A Cookson.
61. S Timoshenko, 'Vibration problems in engineering'.
62. J L Krahula, 'Out-of-plane vibrations of uniform circular ring', Jnl of Applied Mechanics, Sept 1966.
63. R W Brown, 'Centrifugal pump impeller design: mechanical design and cavitation investigation', M Sc thesis, School of Mechanical Engineering, Cranfield Institute of Technology, 1981.

APPENDIX - A

VIBRATION ANALYSIS OF
CYCLICALLY REPEATED STRUCTURES

The method of cyclic sub-substructures provides an effective means for vibration analysis of cyclically repeated structures. The basic input for the analysis comprises the stiffness and mass matrices for the repeated sub-structure.

The general form of the complete stiffness and mass matrices of a typical structure, consisting of four repeated sub-structures is shown in Fig A.1. Non-zero coefficients occur in the shaded areas only and the overlapping sub-matrices are identical. A typical sub-matrix contains N_i internal variables and N_b boundary variables as shown in the figure. The form of these matrices is such that a Fourier transformation may be performed on the variables to convert the matrices into the form shown in Fig A.2. Decoupling between the sets of variables takes place and the new non-zero sub-matrices are each functions of the original sub-structure matrices. The eigensolution can now be performed on one of the decoupled matrices. Details of the Fourier method are described in the following.

A cyclically repetitive structure is shown in Fig A.3, of which Fig A.4 represents a typical sector. The sector boundaries need not be on radial lines, nor do the sectors need to be symmetric. The vector $\bar{V}_i$ represents the displacements of the internal nodes of the i th sector and V_i the displacements of the nodes on the $i-1$ th and i th sector boundary.

If we consider a typical sector, let V_0, V_1 and V_2 be the displacement vectors for the internal, first and second boundary nodes respectively. The stiffness matrix for a repeating sector can be written in the partitioned form as

$$\begin{array}{ccc} K_{11} & K_{10} & K_{12} \\ K_{01} & K_{00} & K_{02} \\ K_{21} & K_{20} & K_{22} \end{array} \quad \dots \quad (A.1)$$

Since every sector is identical the assembled stiffness matrix can be written in the following form, which represents the assembly for only a part of the total structure.

<u>Vi-2</u>	<u>Vi-1</u>	<u>Vi-1</u>	<u>Vi</u>	<u>Vi</u>	<u>Vi+1</u>	<u>Vi+1</u>	
K10	K12						
K00	K02						
K20	K11+K22	K10	K12				
	K01	K00	K02				
	K21	K20	K11+K22	K10	K12		
			K01	K00	K02		
			K21	K20	K11+K22	K10	
					K01	K00	
					K21	K20	. . . (A.2)

The two rows of the assembled matrix outlined above form the cyclically repeating segments of assembled matrices for the i th sector and boundary between the i th and $i-1$ th sectors.

The response (mode shapes) of the system will consist of discrete harmonic variations of a reference mode shape. In performing this decomposition of V_i into harmonics, the number of harmonics required for a structure with p sectors is M , which is $p/2+1$ for p even and $(p+1)/2$ for p odd.

$$V_i = \sum_{n=0}^M U_c^n \cos(2n\pi/p) + U_s^n \sin(2n\pi/p) \quad \dots \quad (A.3)$$

$$\underline{V}_i = \sum_{n=0}^M \underline{U}_c^n \cos(2n\pi/p) + \underline{U}_s^n \sin(2n\pi/p) \quad \dots \quad (A.4)$$

Where U_c and U_s denote the amplitudes for Cosine and Sine components for the n th harmonic on the boundary between the i th and $i-1$ th sectors.

By trigonometric manipulations the expressions for V_{i+1} and V_{i-1} are obtained as,

$$V_{i+1} = \sum_0^M \left[\left(\frac{\cos 2n\pi}{p} \cdot \frac{\cos 2n\pi}{p} \pm \frac{\sin 2n\pi}{p} \cdot \frac{\sin 2n\pi}{p} \right) \{U_c^n\} + \left(\frac{\cos 2n\pi}{p} \cdot \frac{\sin 2n\pi}{p} \mp \frac{\sin 2n\pi}{p} \cdot \frac{\cos 2n\pi}{p} \right) \{U_s^n\} \right] \dots \quad (A.5)$$

$$\underline{V}_{i-1} = \sum_0^M \left[\left(\frac{\cos 2n\pi}{p} \cdot \frac{\cos 2n\pi}{p} + \frac{\sin 2n\pi}{p} \cdot \frac{\sin 2n\pi}{p} \right) \{U_c^n\} + \left(\frac{\cos 2n\pi}{p} \cdot \frac{\sin 2n\pi}{p} - \frac{\sin 2n\pi}{p} \cdot \frac{\cos 2n\pi}{p} \right) \{U_s^n\} \right] \dots \quad (A.6)$$

Substituting the above expressions for V_i , $\underline{V}_i$, V_{i+1} and $\underline{V}_{i-1}$ into the two rows of assembled matrix, representing the cyclically repeating segment, in Eqn (A.2) yields two expressions for the n th harmonic. On uncoupling each expression into two groups belonging to $\cos(2n\pi/p)$ and $\sin(2n\pi/p)$ terms, it will yield four expressions which can be written into the matrix form as shown in Eqn (A.7), with the associated displacement vector $\{U_c \ U_s \ \underline{U}_c \ \underline{U}_s\}^T$.

$$\begin{bmatrix} K_{11}+K_{22}+(K_{21}+K_{12}) \cdot C & -(K_{21}-K_{12}) \cdot S & K_{10}+K_{20} \cdot C & -K_{20} \cdot S \\ (K_{21}-K_{12}) \cdot S & K_{11}+K_{22}+(K_{21}+K_{12}) \cdot C & K_{20} \cdot S & K_{10}+K_{20} \cdot C \\ K_{01}+K_{02} \cdot C & K_{02} \cdot S & K_{00} & 0 \\ -K_{02} \cdot S & K_{01}+K_{02} \cdot C & 0 & K_{00} \end{bmatrix} \quad (A.7)$$

where $C = \frac{\cos 2n\pi}{P}$, $S = \frac{\sin 2n\pi}{P}$

In the above matrix, the first two rows correspond to the $\cos(2n\pi/p)$ and $\sin(2n\pi/p)$ terms respectively, which are obtained from the first of the two rows of the assembled matrix in Eqn (A.2). Similarly, the third and fourth rows above correspond to the later row of the assembled matrix.

The matrix in Eqn (A.7) now represents the uncoupled repeated stiffness matrix which is characteristic of the complete stiffness matrix for each harmonic. It can be written in brief notation as $[\bar{K}_n]\{\bar{U}_n\}$.

Similar process on the mass matrix yields $[\bar{M}_n]\{\bar{U}_n\}$, where $[\bar{M}_n]$ has the same form as $[\bar{K}_n]$.

The vibration problem now reduces to the determination of the eigenvalues and the eigenvectors of the reduced system,

$$([\bar{K}_n] - \omega^2 [\bar{M}_n])\{\bar{U}_n\} = 0 \quad \dots \dots (A.8)$$

In the above case, for a sector with N_i internal and N_b boundary degrees of freedom, the order of the system equation would be $2(N_i+N_b)$. The factor 2 reflects the phase and quadrature components of the motion. On eigensolution, the system leads to two sets of identical eigenvalues, one each from the phase and quadrature components. However, the eigenvectors of the two sets representing the phase and the quadrature components of the displacements will be different.

In practice, the stiffness and mass matrices are built up and the motion is solved for each harmonic (ie nodal diametral mode) of the system independently. In general, a typical structure with repeatable substructures represent a fairly large order of eigenvalue problem. The above procedure leads to economisation as the eigenvalue problem is now solved for a reduced system representing the size of only one substructure.

Complex Eigen Solution

Further reduction in computer core and time can be achieved, for the above solution, by use of an extension to a complex eigensolution.

The associated displacement vector of the decoupled stiffness matrix (Eqn A.7) may be rearranged as $\{\underline{U_c} \ \underline{U_c} \ \underline{U_s} \ \underline{U_s}\}^T$. The rearranged stiffness matrix would be,

$$\begin{bmatrix} K_{00} & K_{01}+K_{02}.C & 0 & K_{02}.S \\ K_{10}+K_{20}.C & K_{11}+K_{22}+(K_{21}+K_{12}).C & -K_{20}.S & -(K_{21}-K_{12}).S \\ 0 & -K_{02}.S & K_{00} & K_{01}+K_{02}.C \\ K_{20}.S & (K_{21}-K_{12}).S & K_{10}+K_{20}.C & K_{11}+K_{22}+(K_{21}+K_{12}).C \end{bmatrix} \quad (A.9)$$

$$\text{Set } \underline{V_c} = \begin{bmatrix} \underline{U_c} \\ \underline{U_c} \end{bmatrix}, \quad \underline{V_s} = \begin{bmatrix} \underline{U_s} \\ \underline{U_s} \end{bmatrix},$$

$$K_A = \begin{bmatrix} K_{00} & K_{01}+K_{02}.C \\ K_{10}+K_{20}.C & K_{11}+K_{22}+(K_{21}+K_{12}).C \end{bmatrix} \text{ and}$$

$$K_B = - \begin{bmatrix} 0 & K_{02}.S \\ -K_{20}.S & -(K_{21}-K_{12}).S \end{bmatrix} \quad \dots \dots \dots (A.10)$$

The stiffness matrix can now be written in the form

$$\begin{bmatrix} K_A & -K_B \\ K_B & K_A \end{bmatrix} \begin{bmatrix} \underline{V_c} \\ \underline{V_s} \end{bmatrix} \quad \dots \dots \dots (A.11)$$

The above matrix can be condensed to a complex matrix of the form $[K_A + jK_B]$ with an associated complex displacement vector $\{\underline{V_c} + j\underline{V_s}\}$. This condensation can be shown by the following simple expansion and comparison.

Eqn (A.11) on expansion gives

$$K_A \underline{V_c} - K_B \underline{V_s} \quad \dots \dots \dots (A.12)$$

$$K_B \underline{V_c} + K_A \underline{V_s} \quad \dots \dots \dots (A.13)$$

The complex form $[K_A + jK_B] \{\underline{V_c} + j\underline{V_s}\}$ gives

$$\underline{\text{real}} \quad K_A \underline{V_c} - K_B \underline{V_s} \quad \dots \dots \dots (A.14)$$

$$\underline{\text{imag}} \quad j(K_B \underline{V_c} + K_A \underline{V_s}) \quad \dots \dots \dots (A.15)$$

An inspection of the above four equations shows the equivalent form. The real dynamic eigenequation, Eqn (A.8), can be now written in complex form as

$$([KA+jKB] - \omega^2 [MA+jMB]) \{Vc+jVs\} = 0 \quad \dots \dots \dots (A.16)$$

It may be written for a solution as,

$$([MA+jMB] [KA+jKB] - \omega^2 [I]) \{Vc+jVs\} = 0 \quad \dots \dots \dots (A.17)$$

There are several special features of the matrices, derived above, which are worth noting. The stiffness matrix in Eqn (A.11) is symmetric as it is derived from a conservative structure. Then $[KA]$ must be symmetric and $[KB]$ skew-symmetric. The complex form $[KA+jKB]$ is known as Hermitian. However, the product matrix $[M^*K]$ is not a Hermitian and requires a general complex eigensolution. In practice, the eigenvalues are algebraically complex but numerically real with the imaginary part typically of order $10E-10$ (for Double Precision arithmetic).

The change to complex method reduces the computer core requirement by 50% and results in a large saving in time, as a reduction of the order of the matrix problem by half easily offsets the expansion due to the use of complex arithmetic.

FIGURES

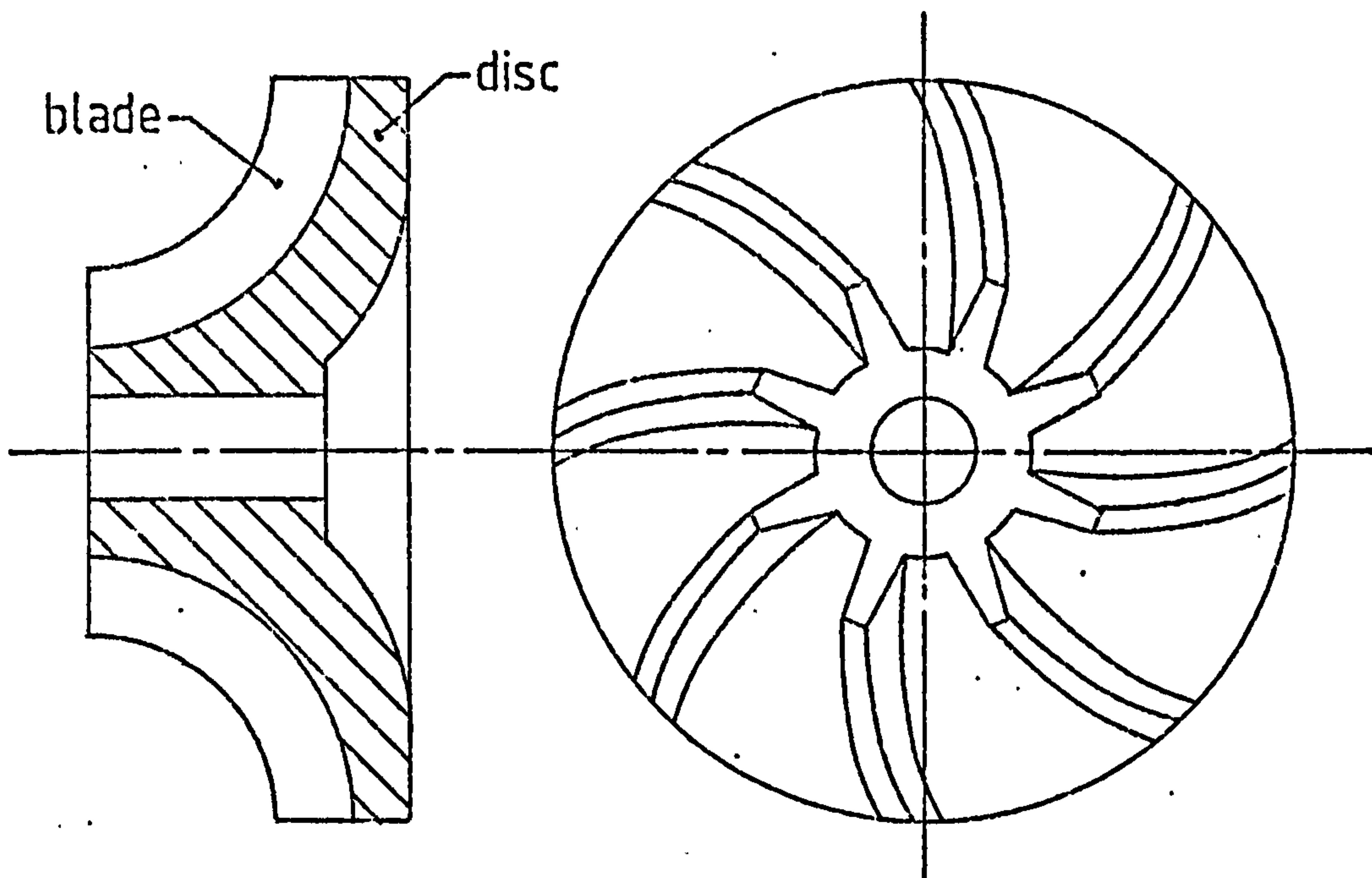


FIG 1-1 A Typical Centrifugal Impeller.

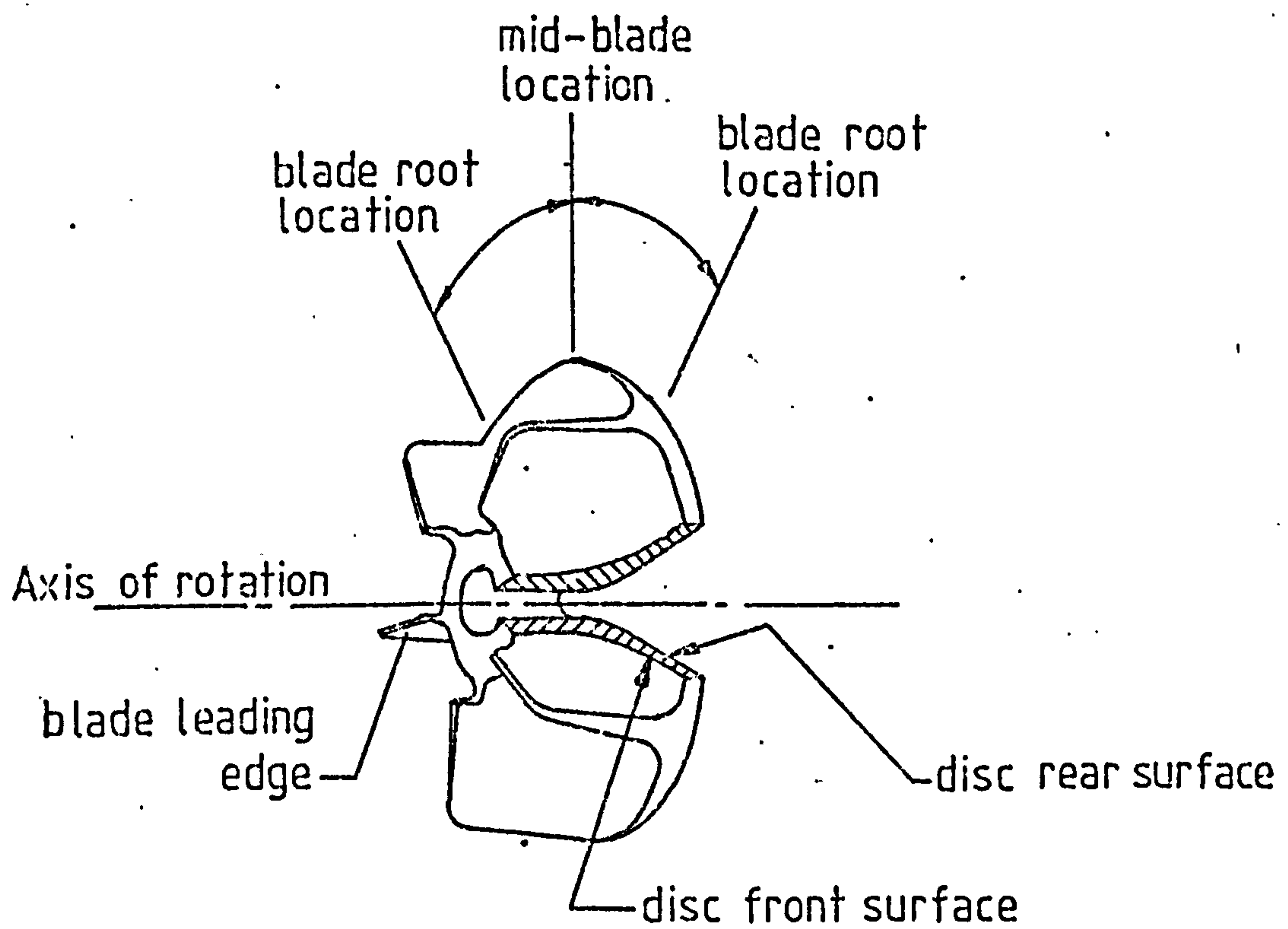


FIG 1-2 Disc Terminology.

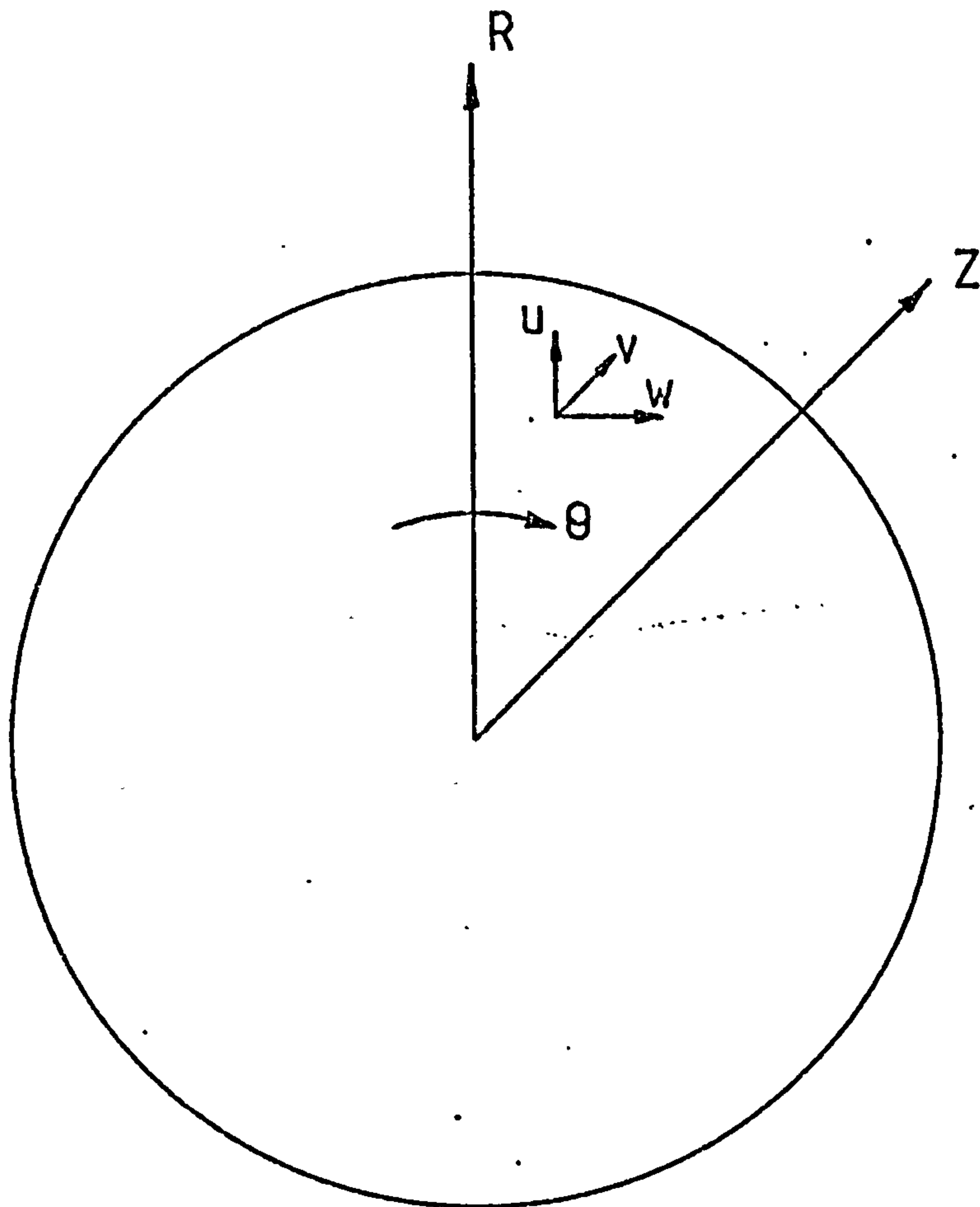


FIG 2-1 Cylindrical System of Co-ordinates.

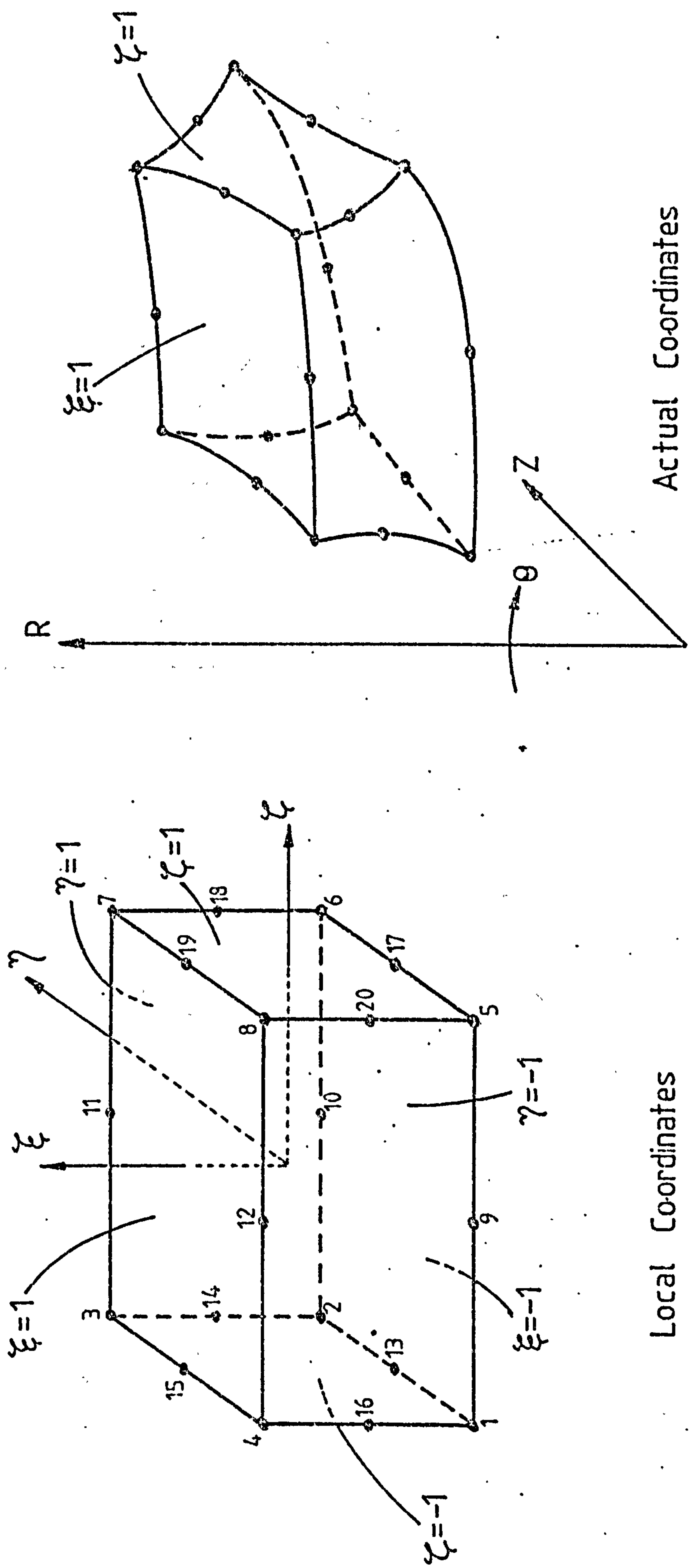


FIG 2-2 20-node Isoparametric Brick Element.

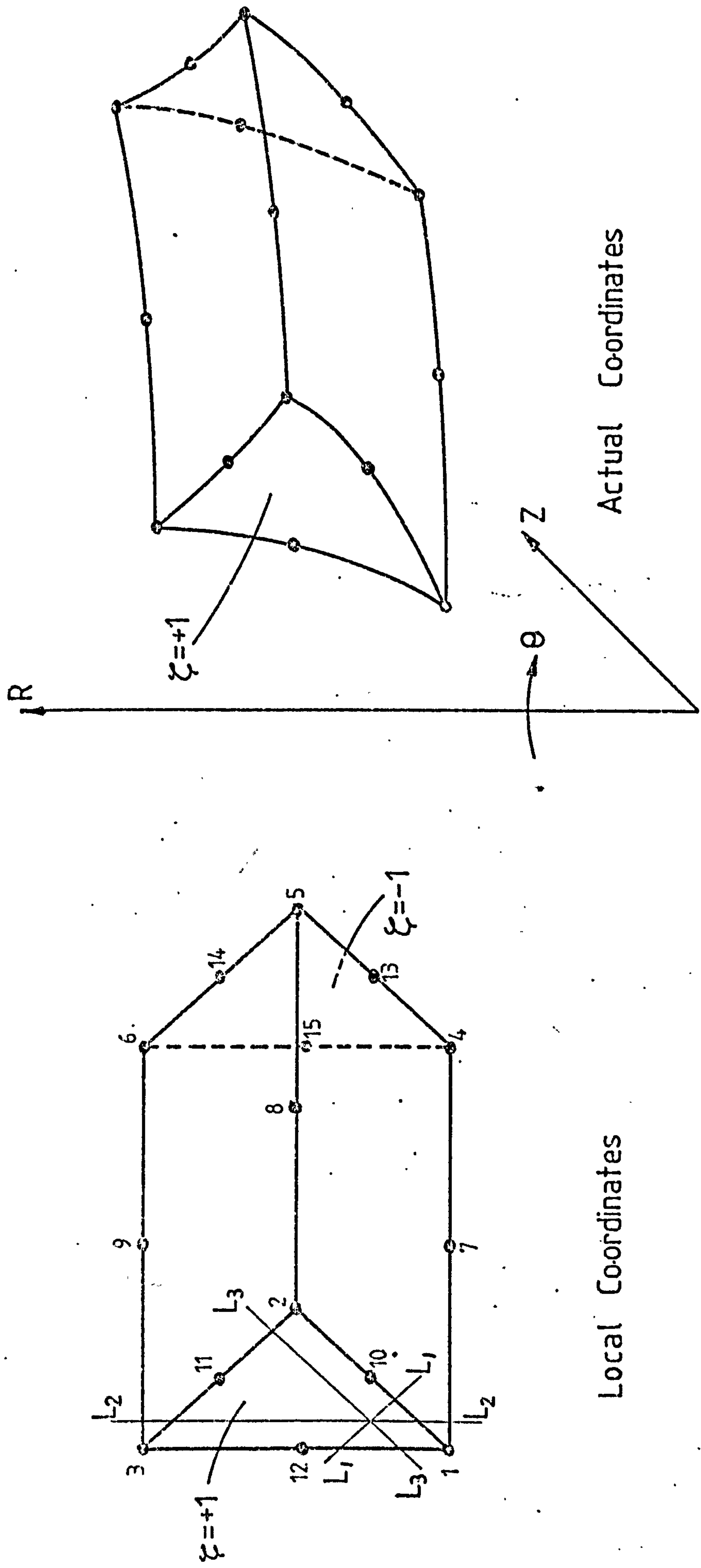


FIG 2-3 15-node Isoparametric Prism Element.

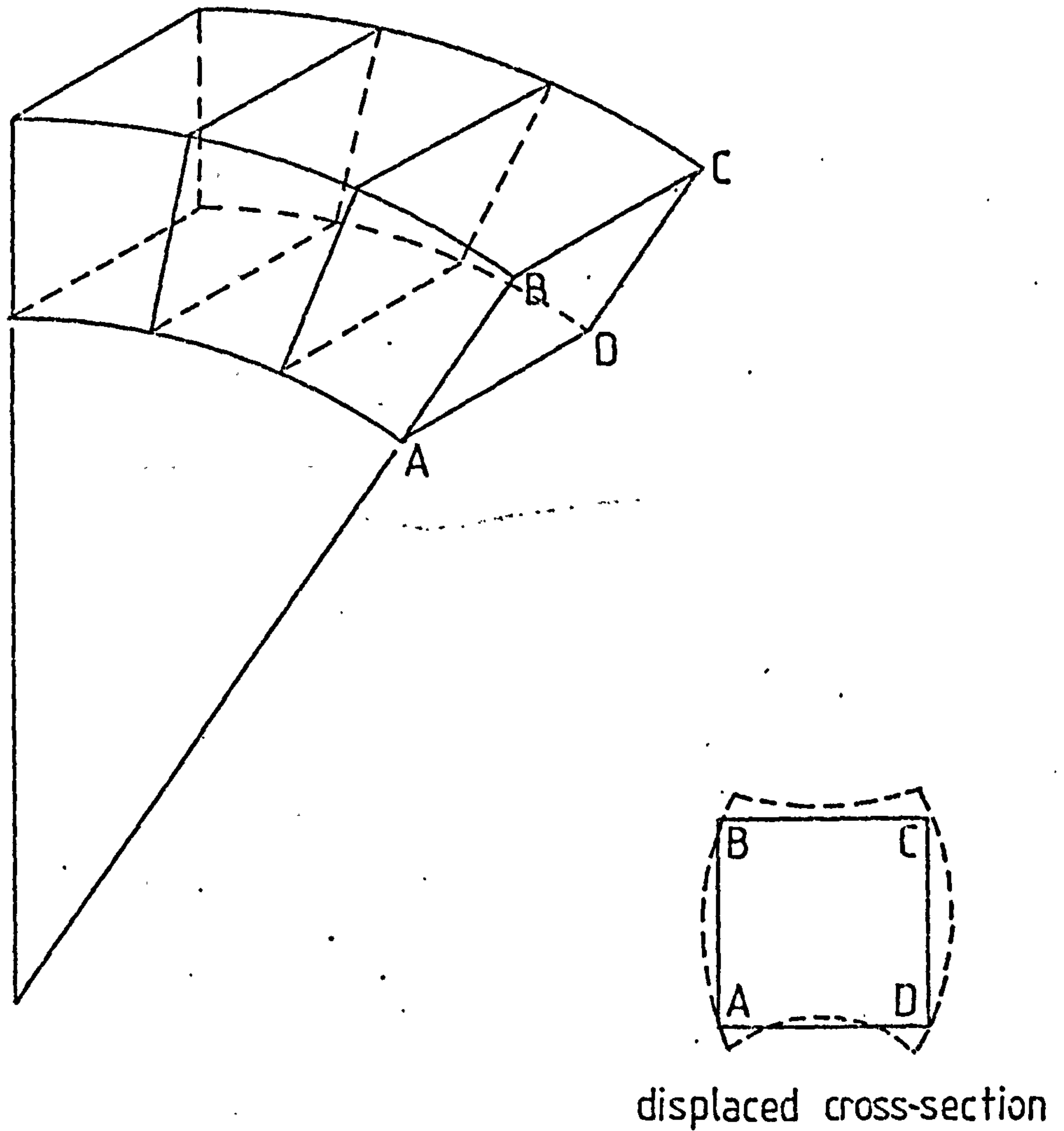


FIG 2-4 A Sector of a Ring (Under Centrifugal Load) Modelled with Three 20-node Elements.

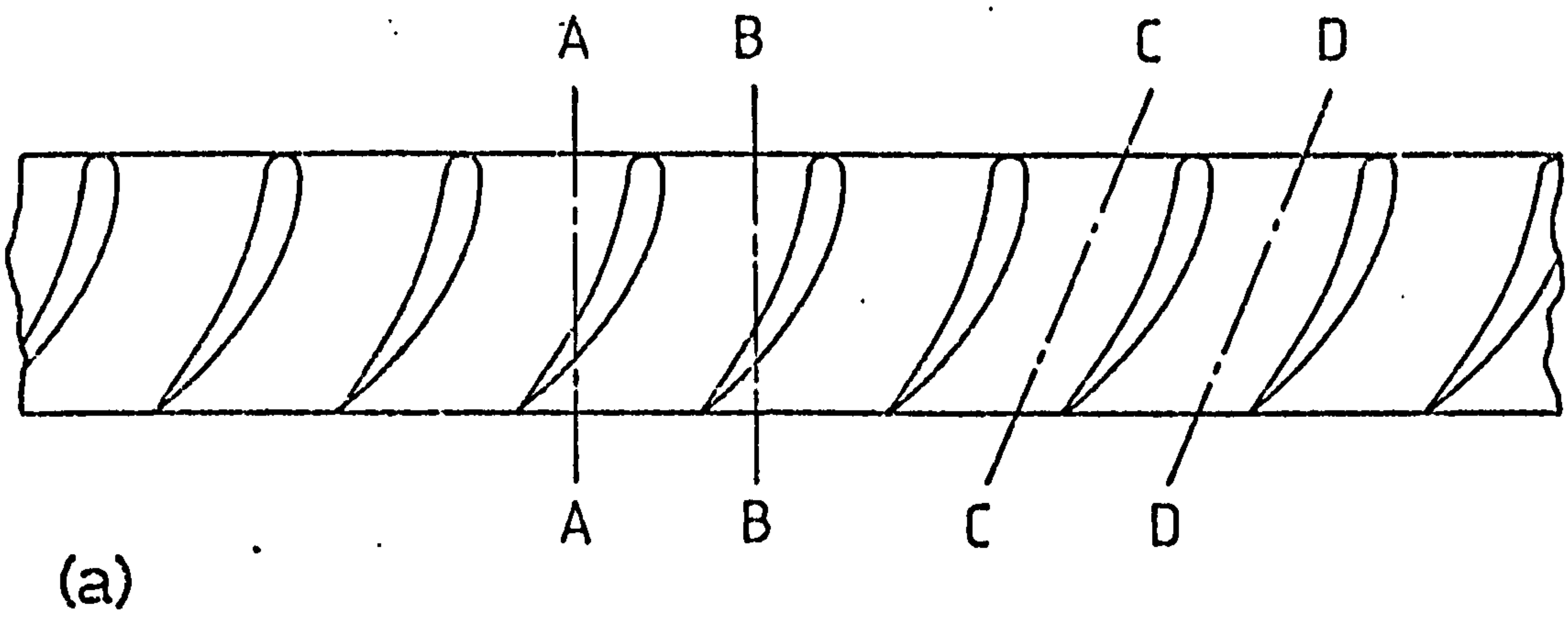
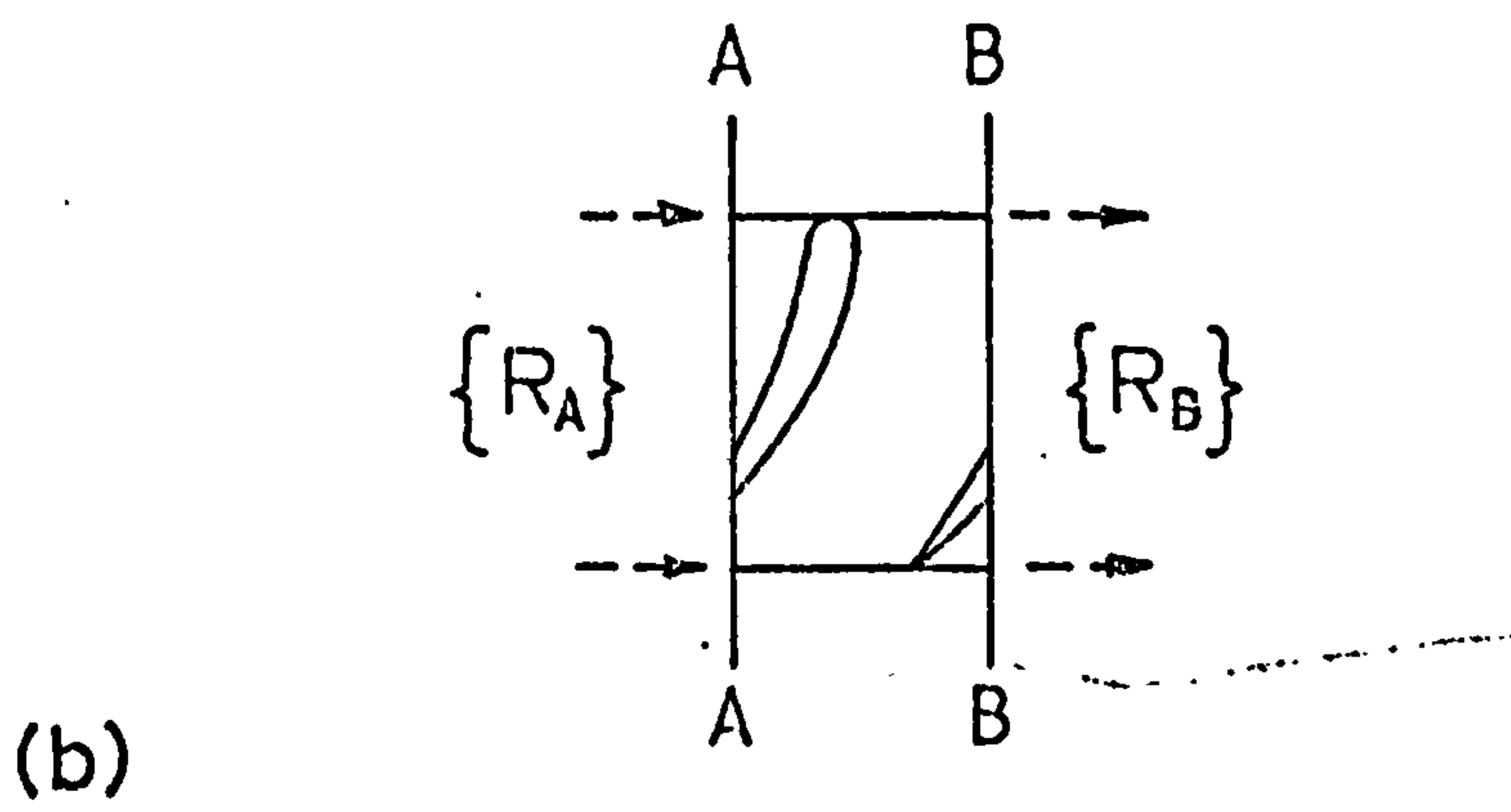
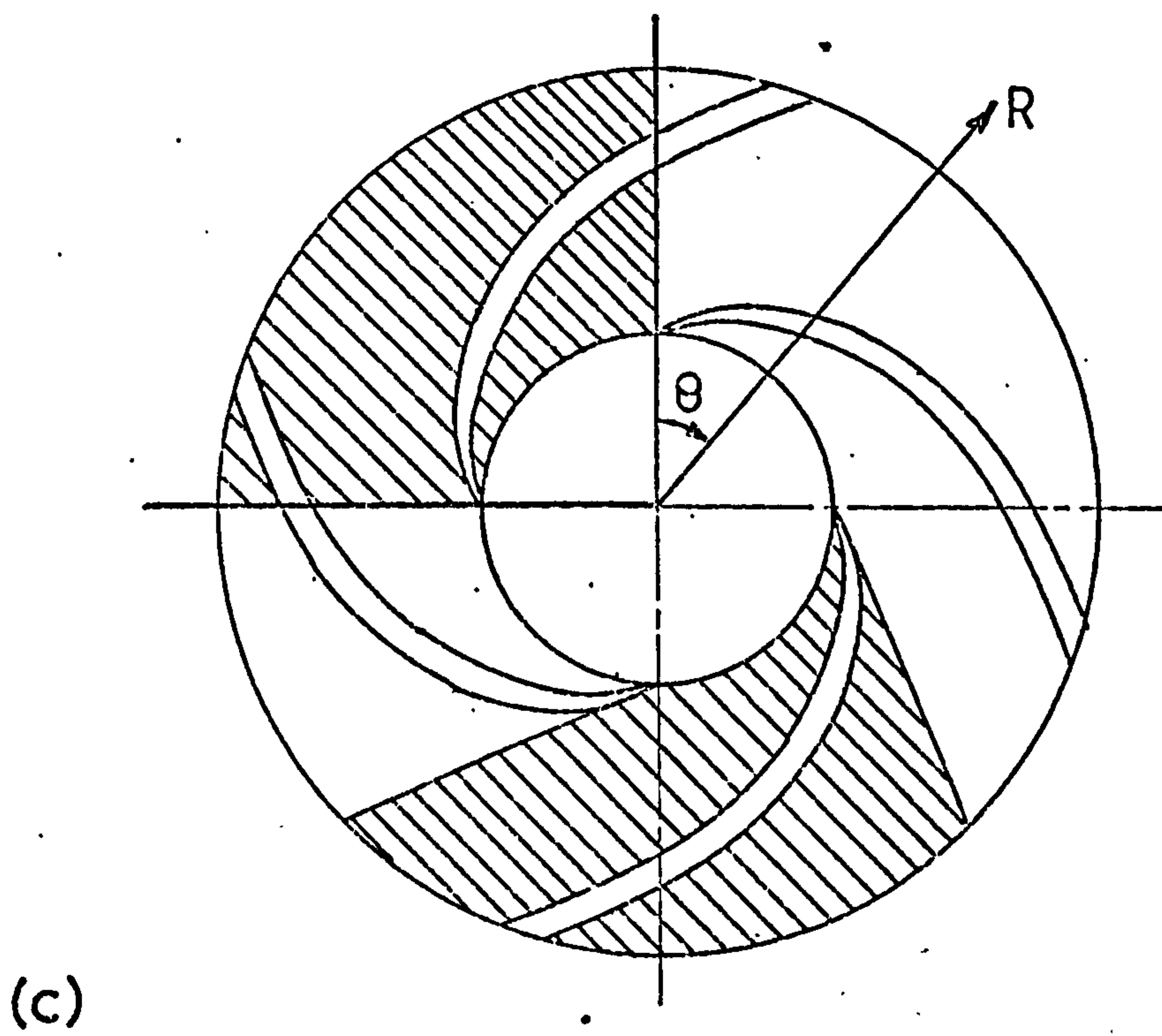


FIG 2-5 Repeatabl Structures.

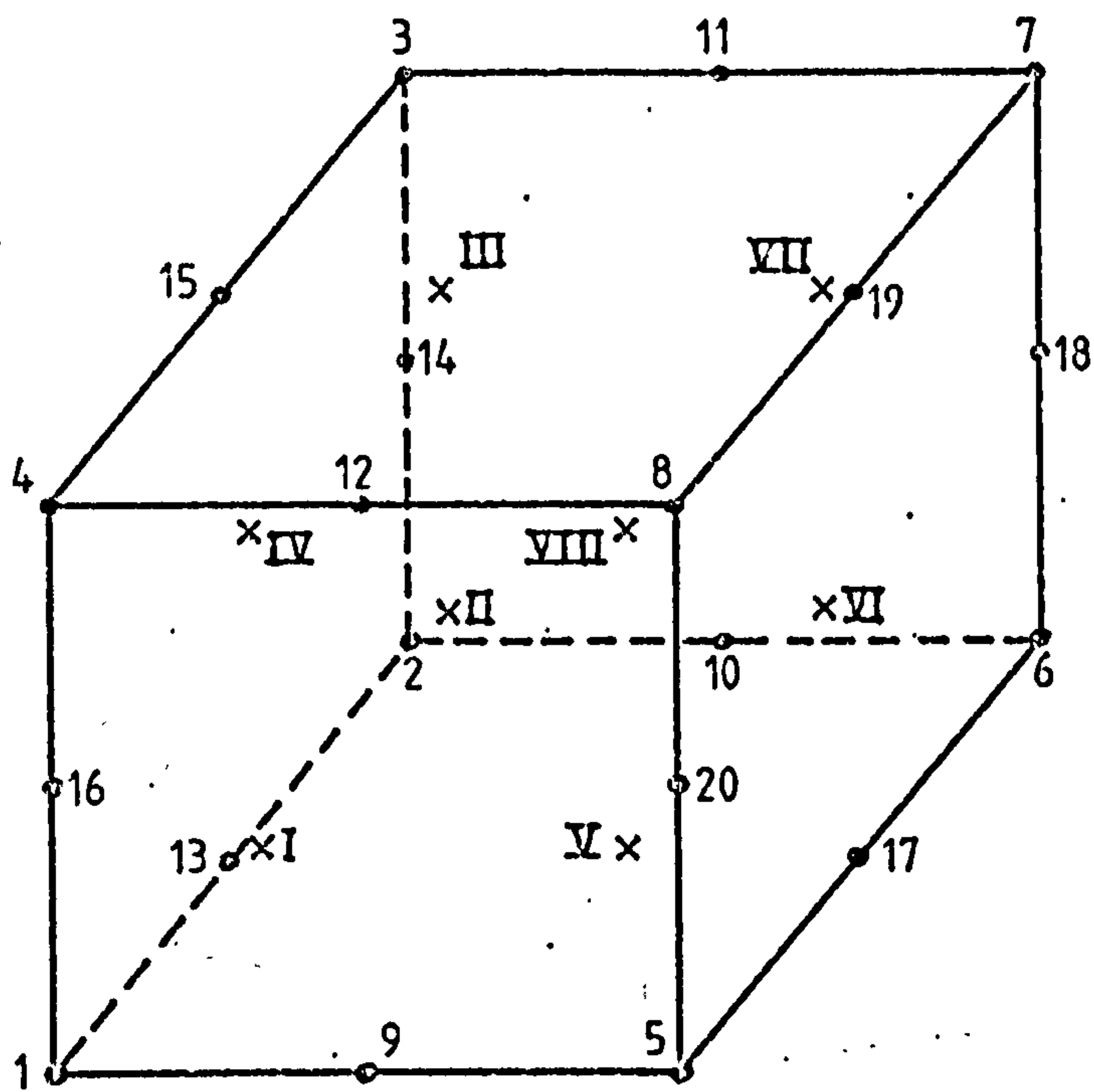


FIG 2-6 Optimal Sampling Points in a 20-node Element.

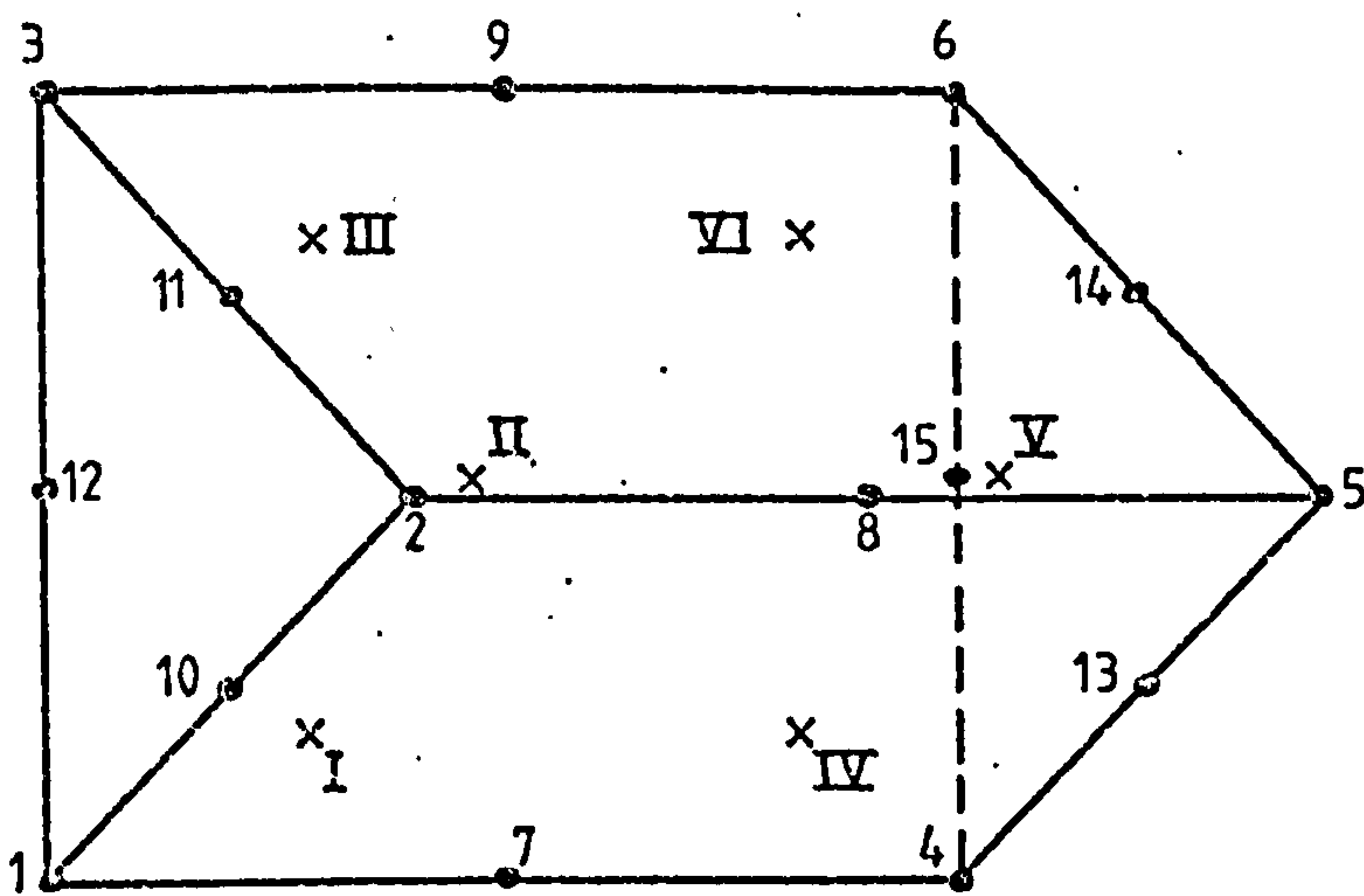


FIG 2-7 Optimal Sampling Points in a 15-node Element.

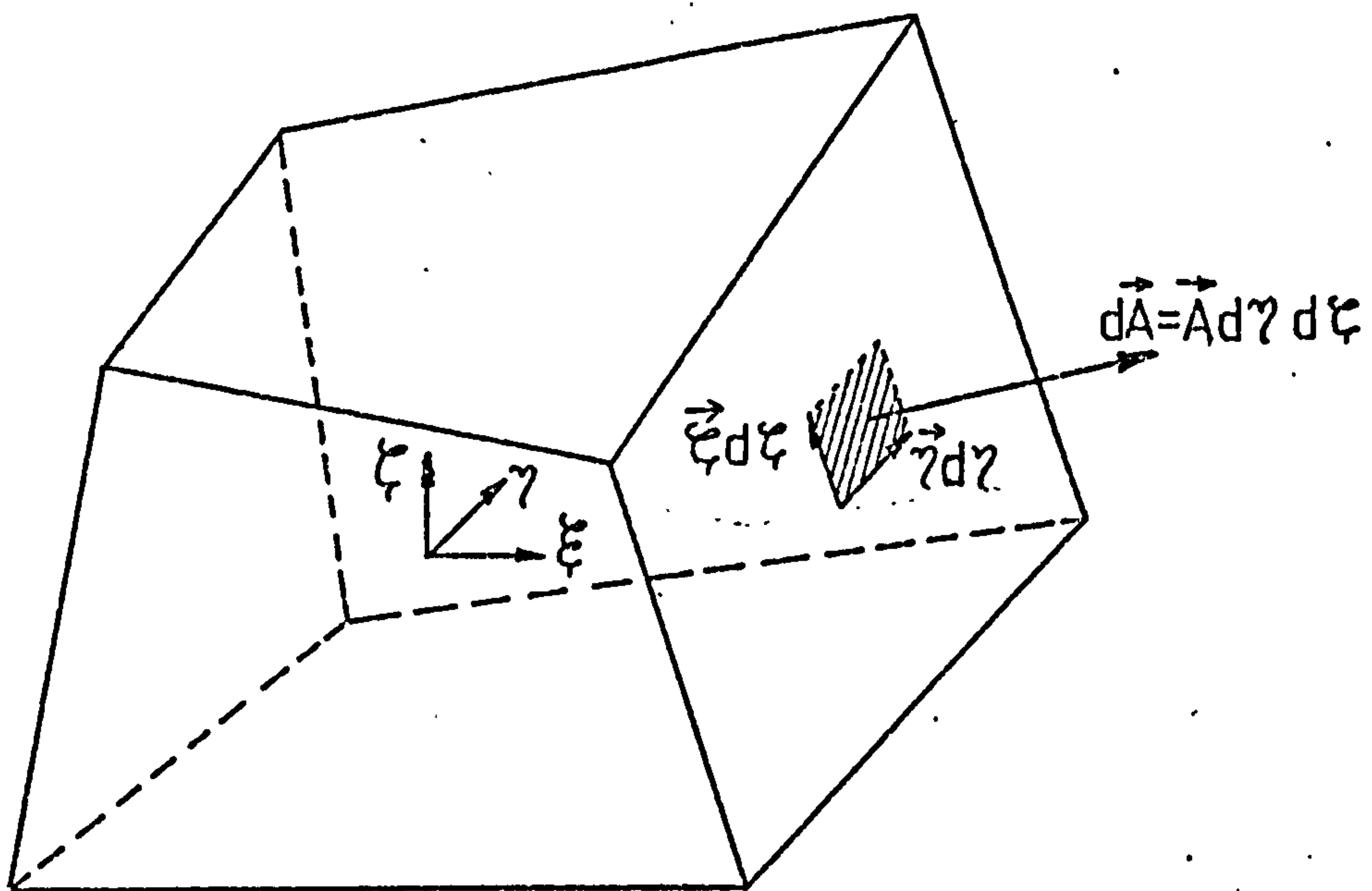


FIG 2-8 A Brick Element Showing an Infinitesimal Surface Area as a Vector Normal to the Face.

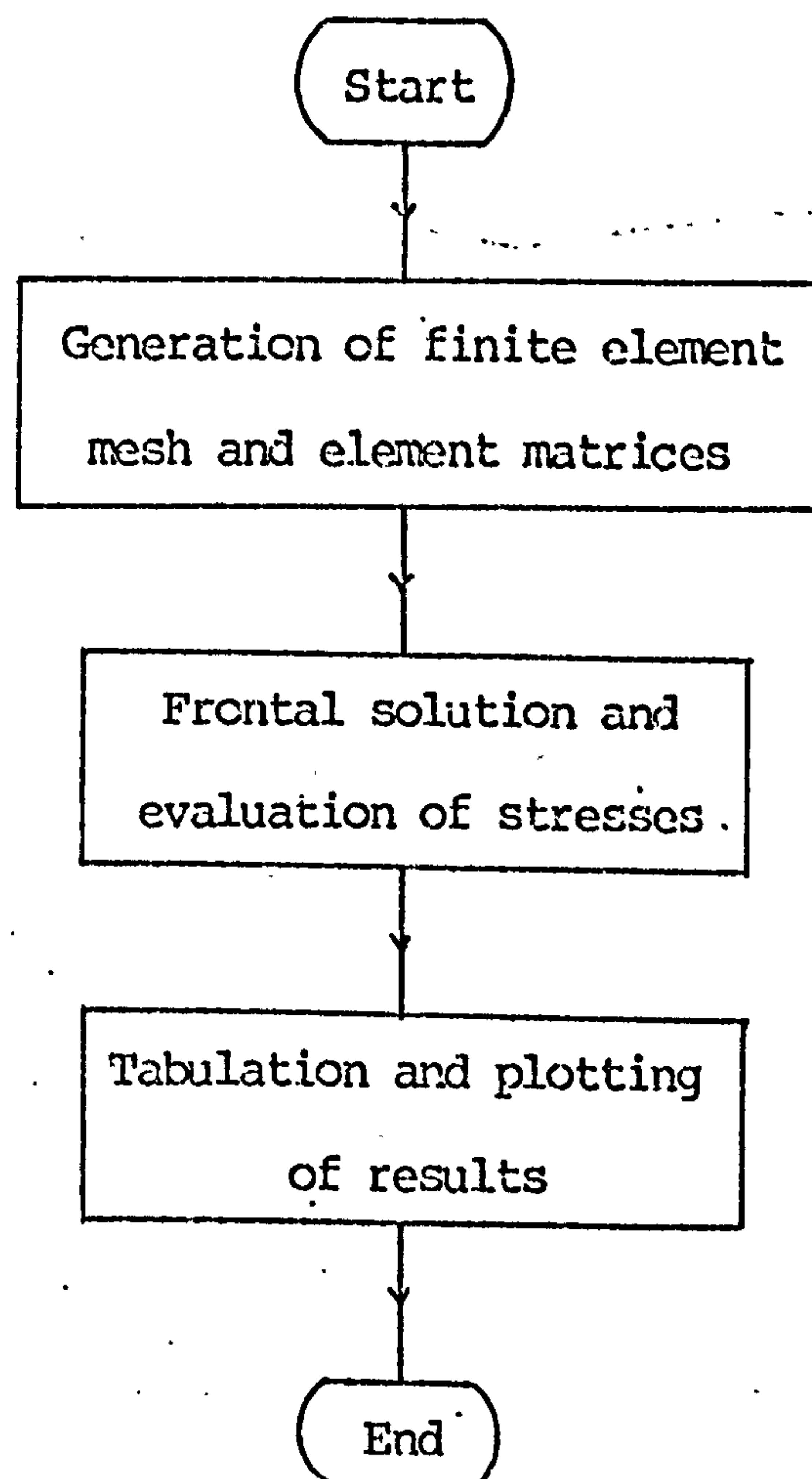


Fig 3.1 Overall flow diagram

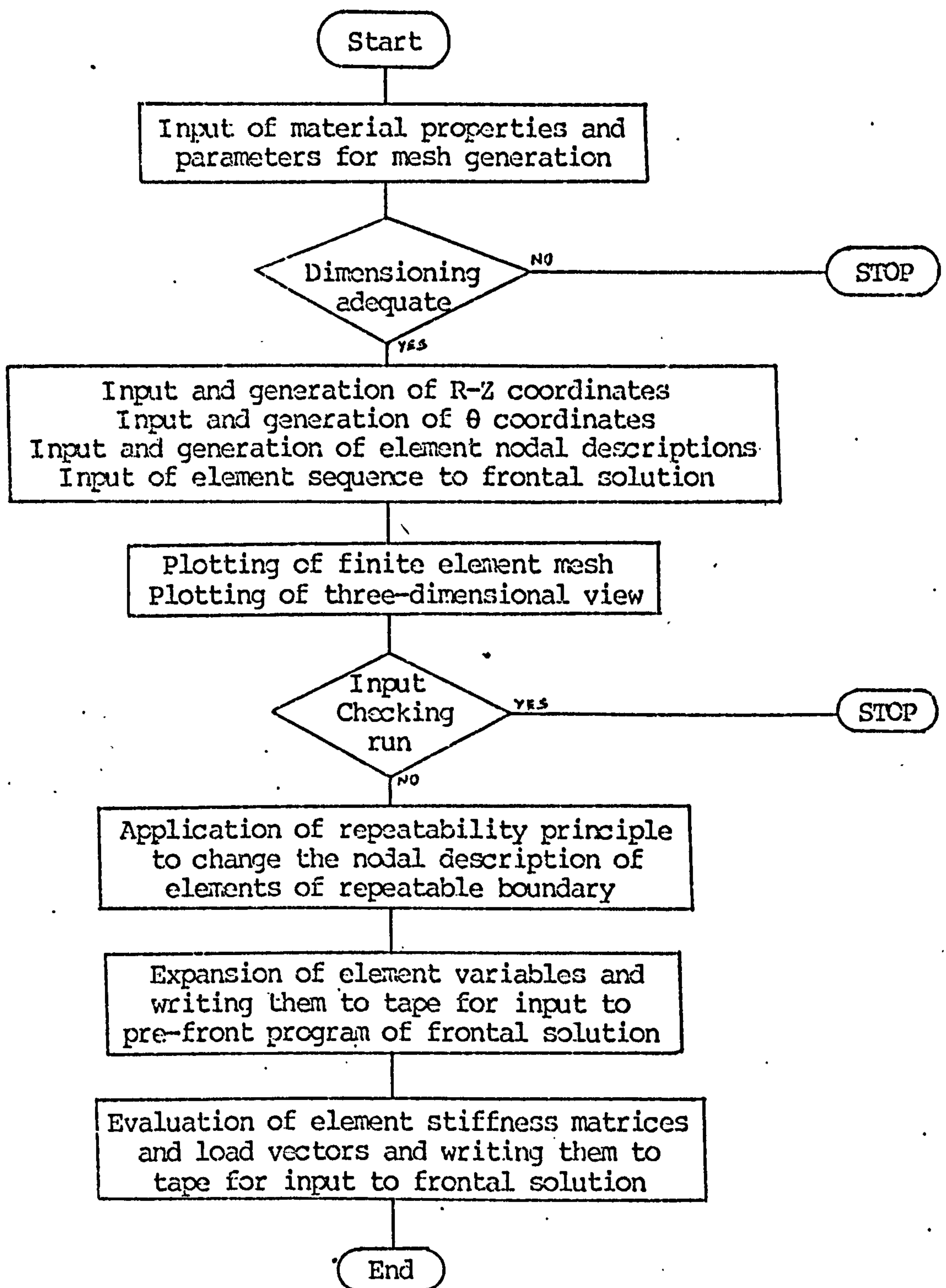
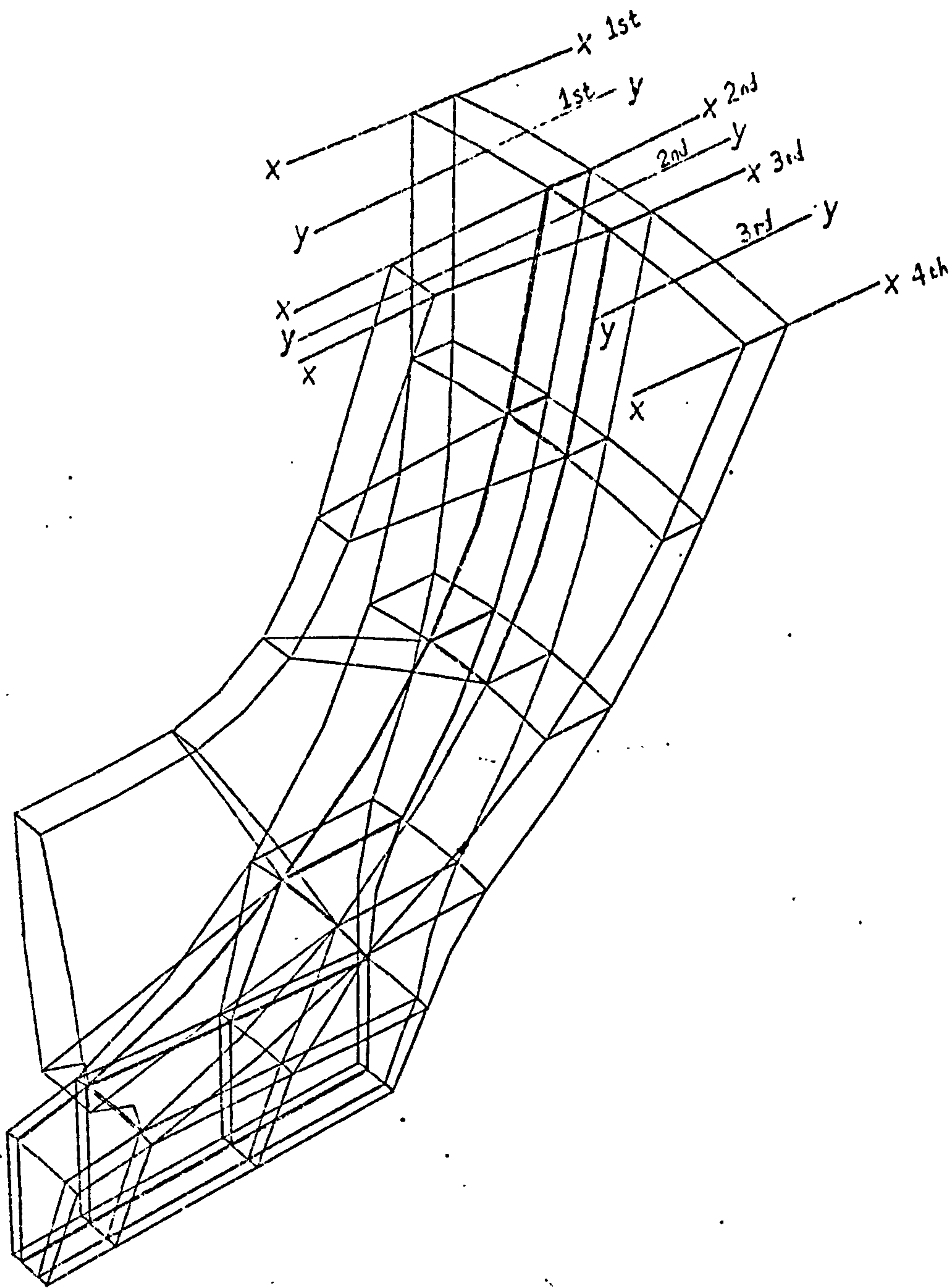


Fig 3.2 Flow diagram of Input Program



3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
 DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS

FIG 3.3(a) IMPELLER MODEL-P MESH-I

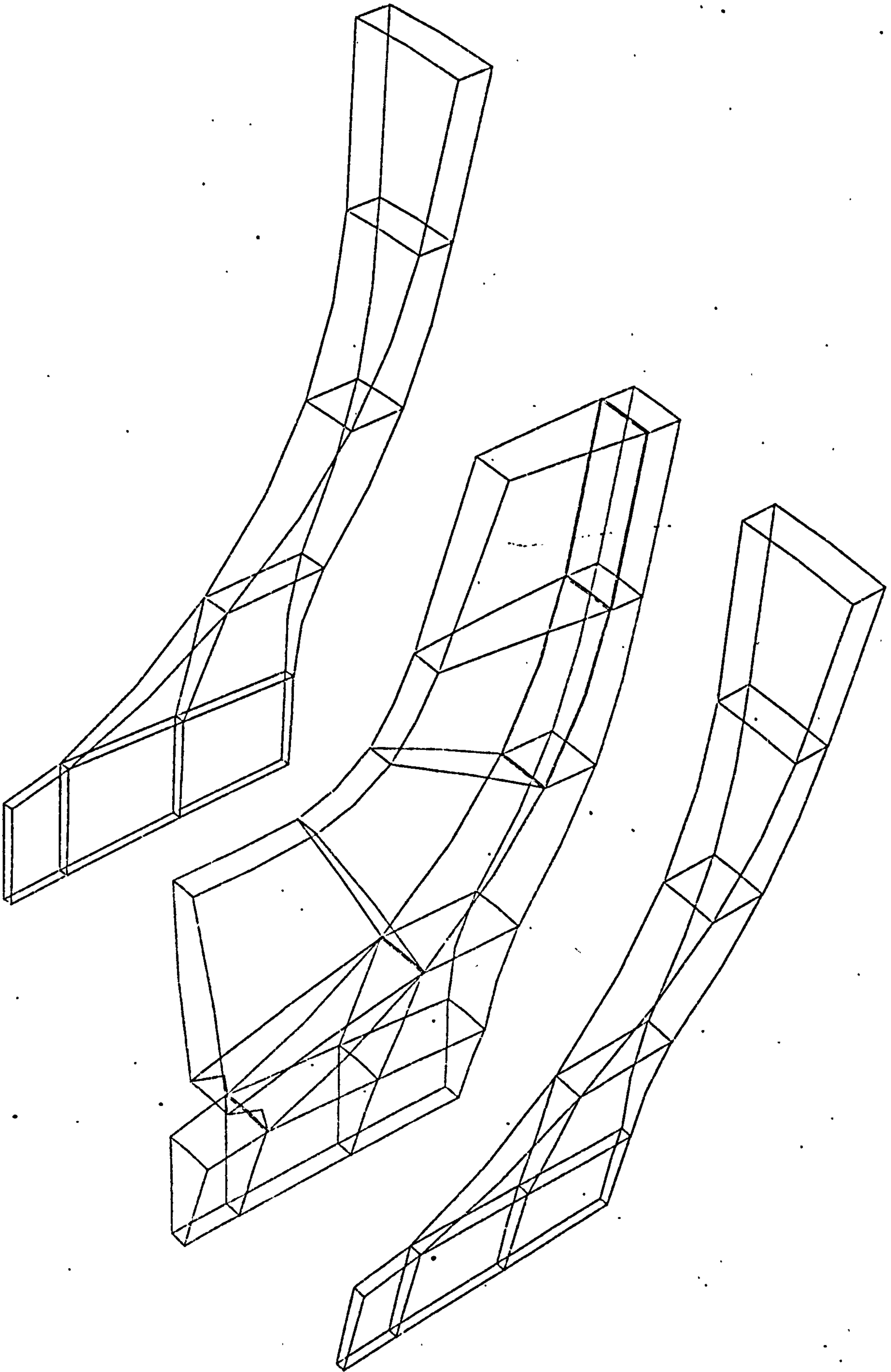
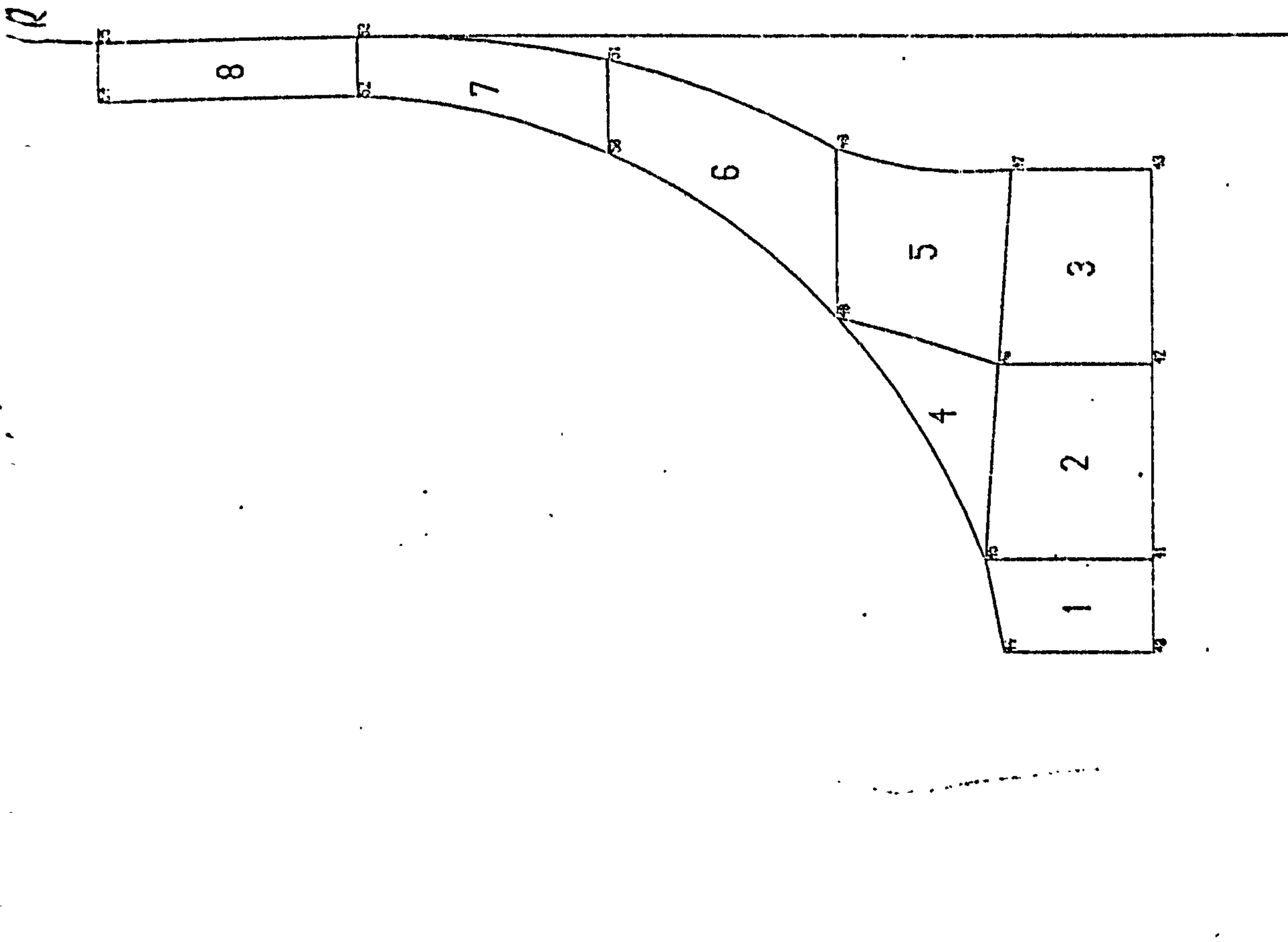


FIG 3-3(b) An Exploded View of IMPELLER MODEL-P MESH-I



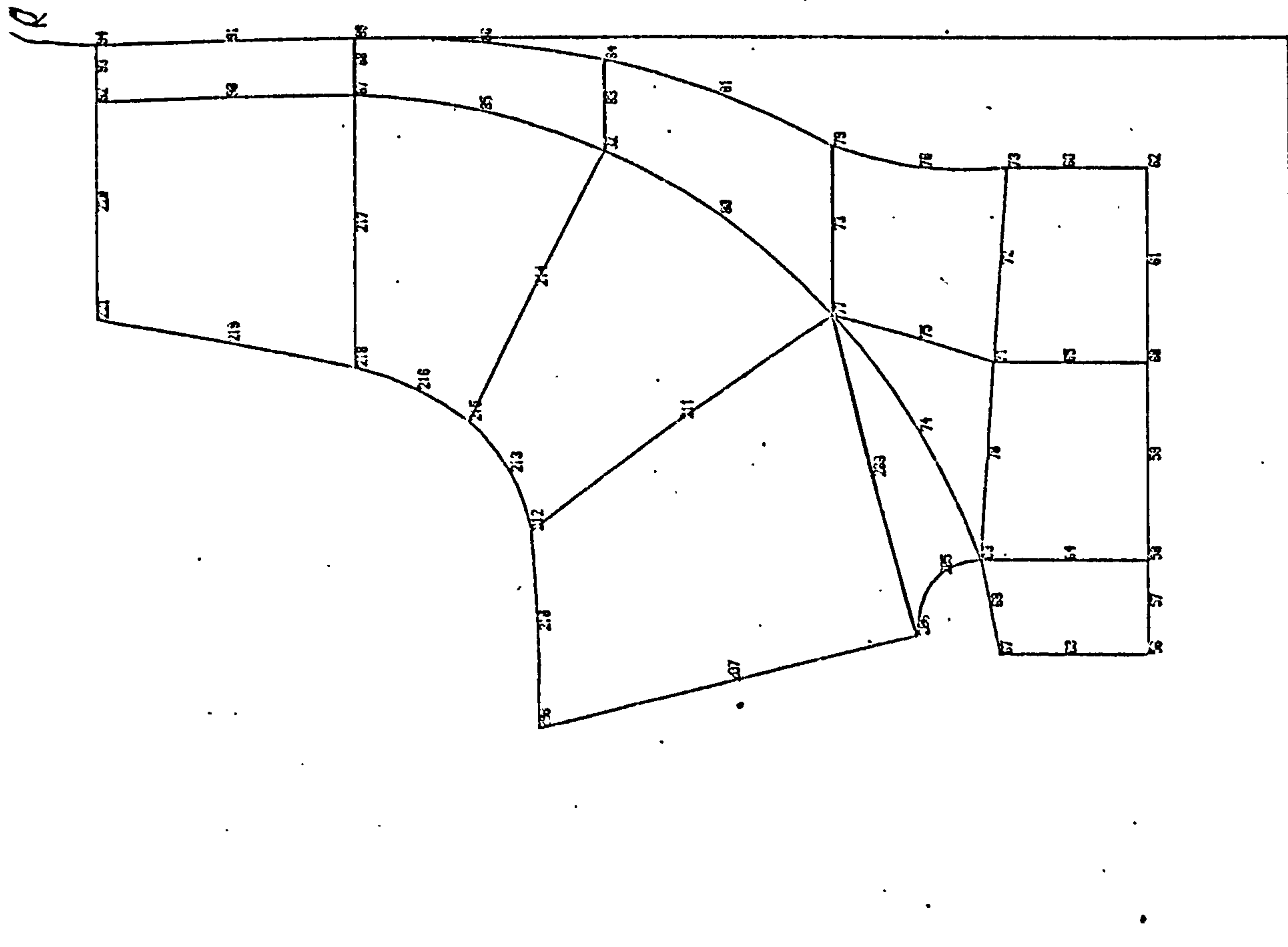
X-X Plane

Y-Y Plane of Layer 1

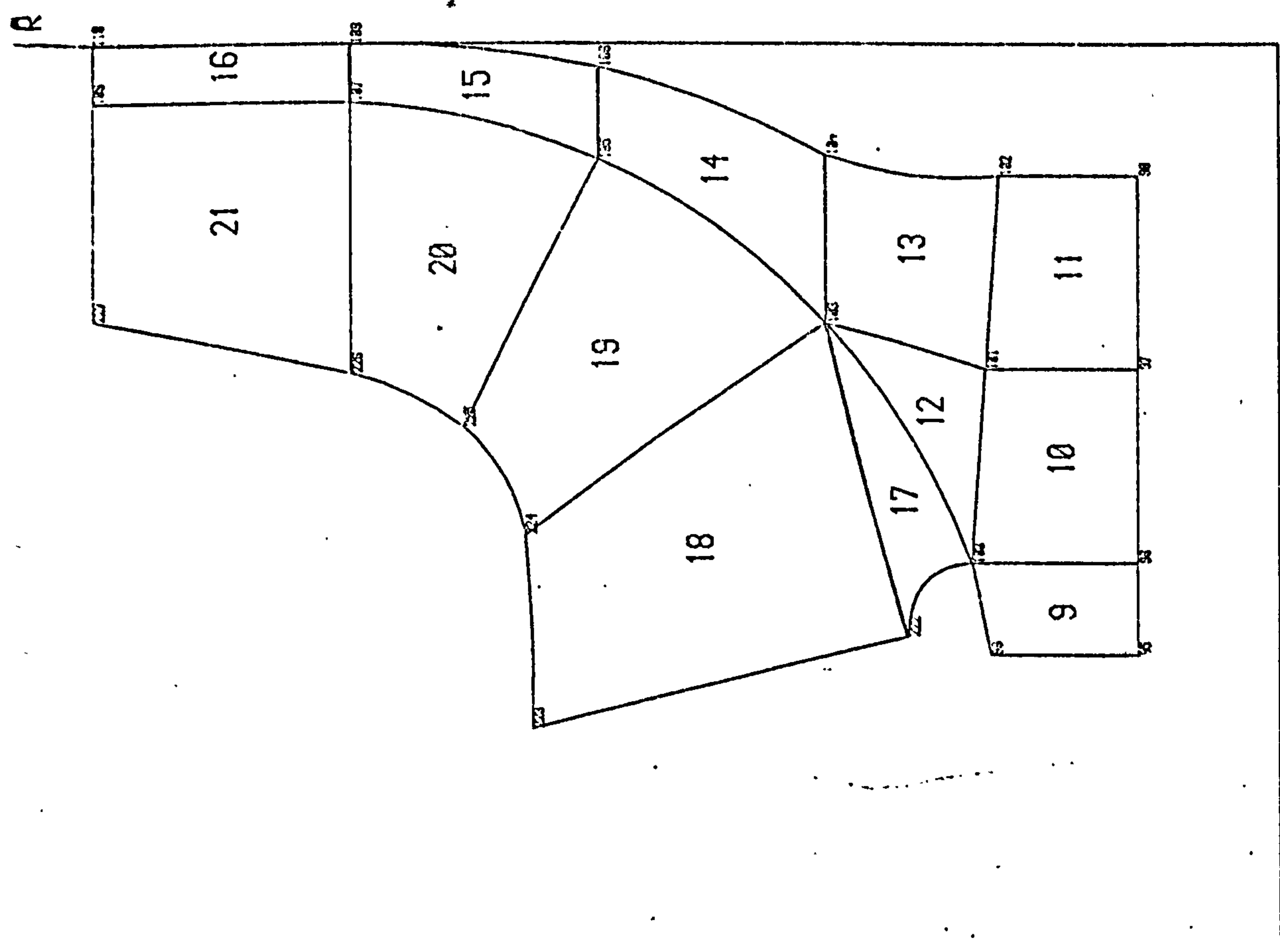
IMPELLER MODEL-P MESH-I

DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS
Sheet 1

FIG 3-3(c)



Z X-X Plane

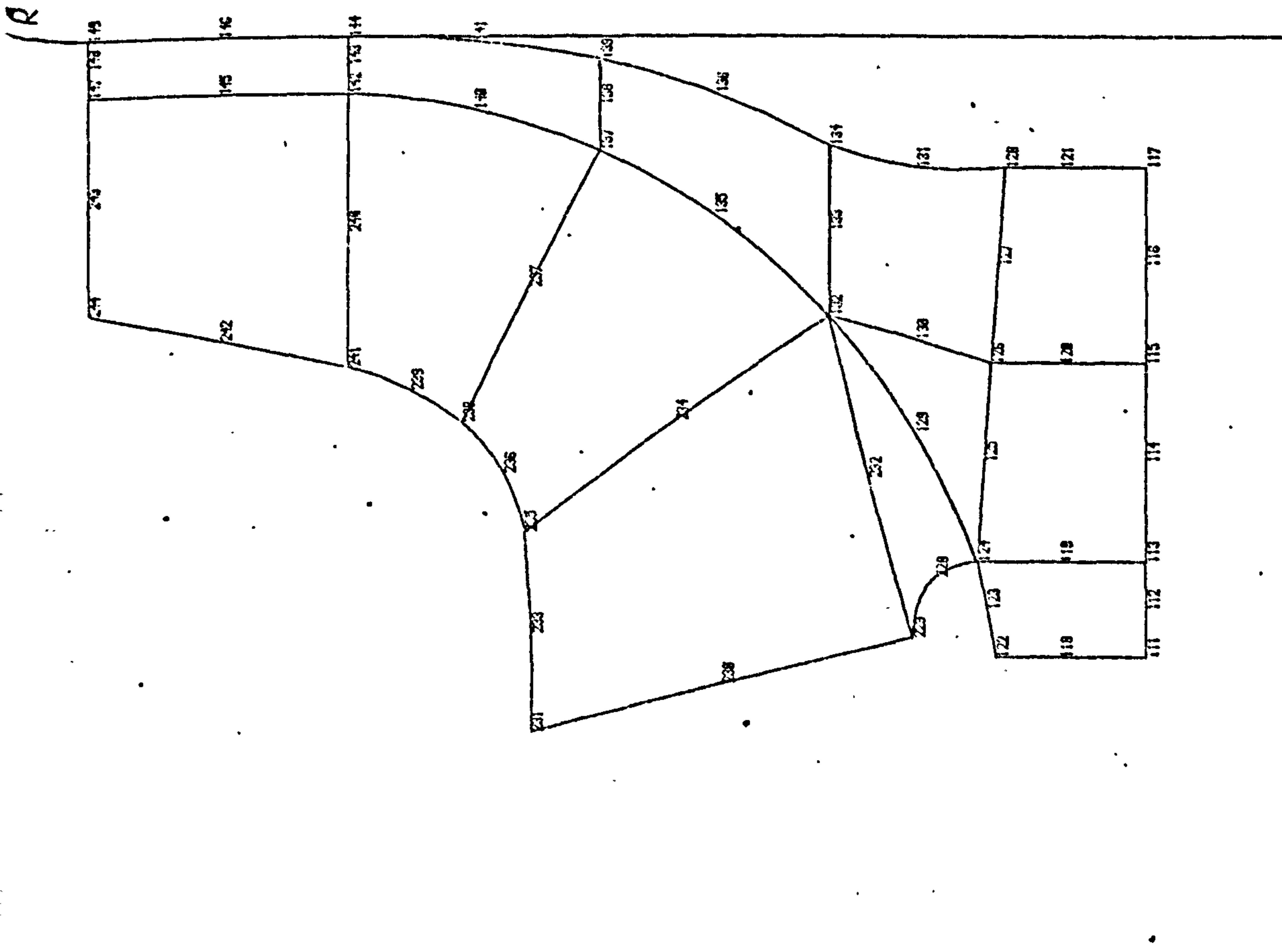


Z Y-Y Plane of Layer 2

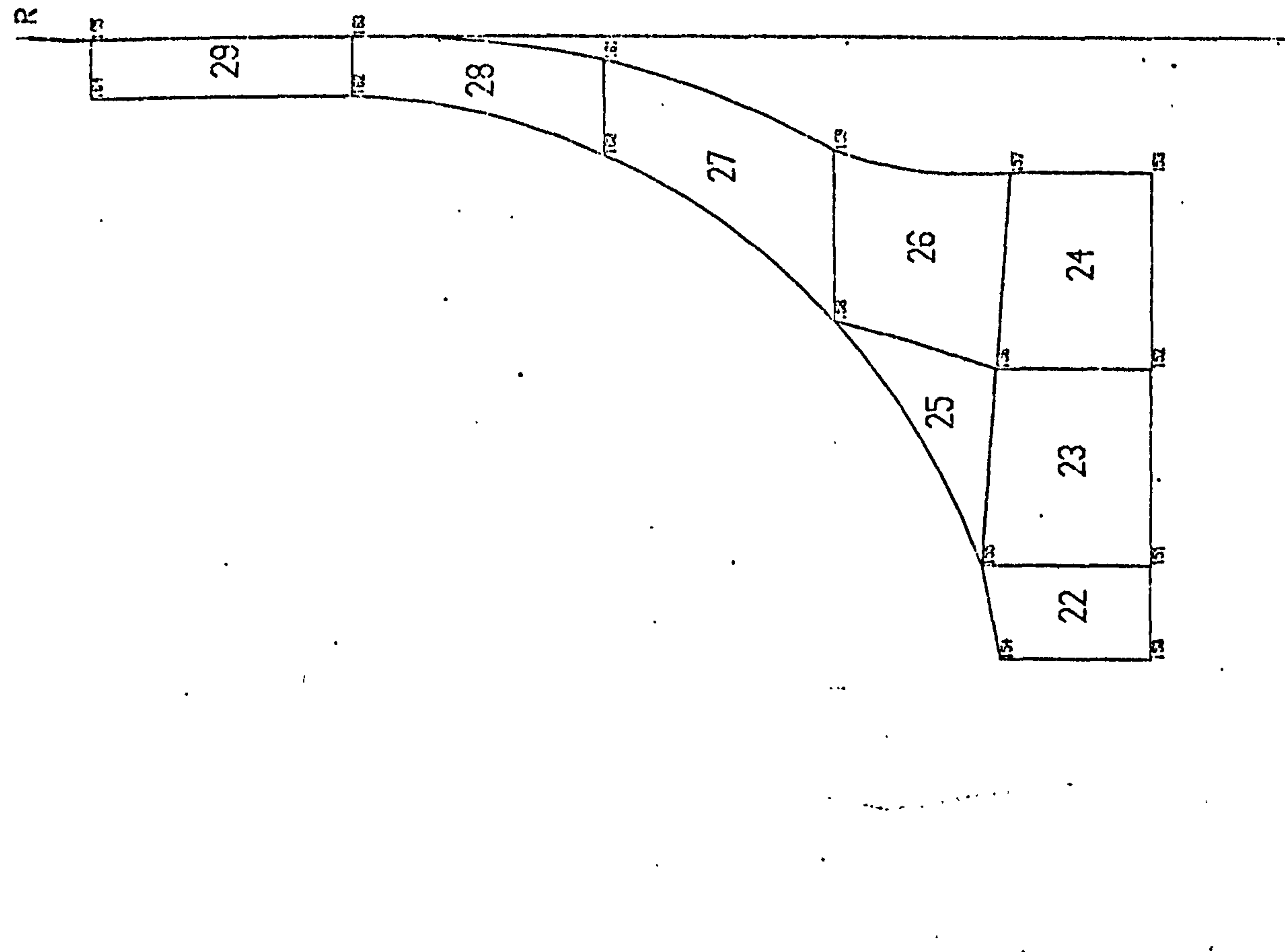
DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS
Sheet 2

IMPELLER MODEL-P MESH-I

FIG 3-3(d)



Z X-X Plane

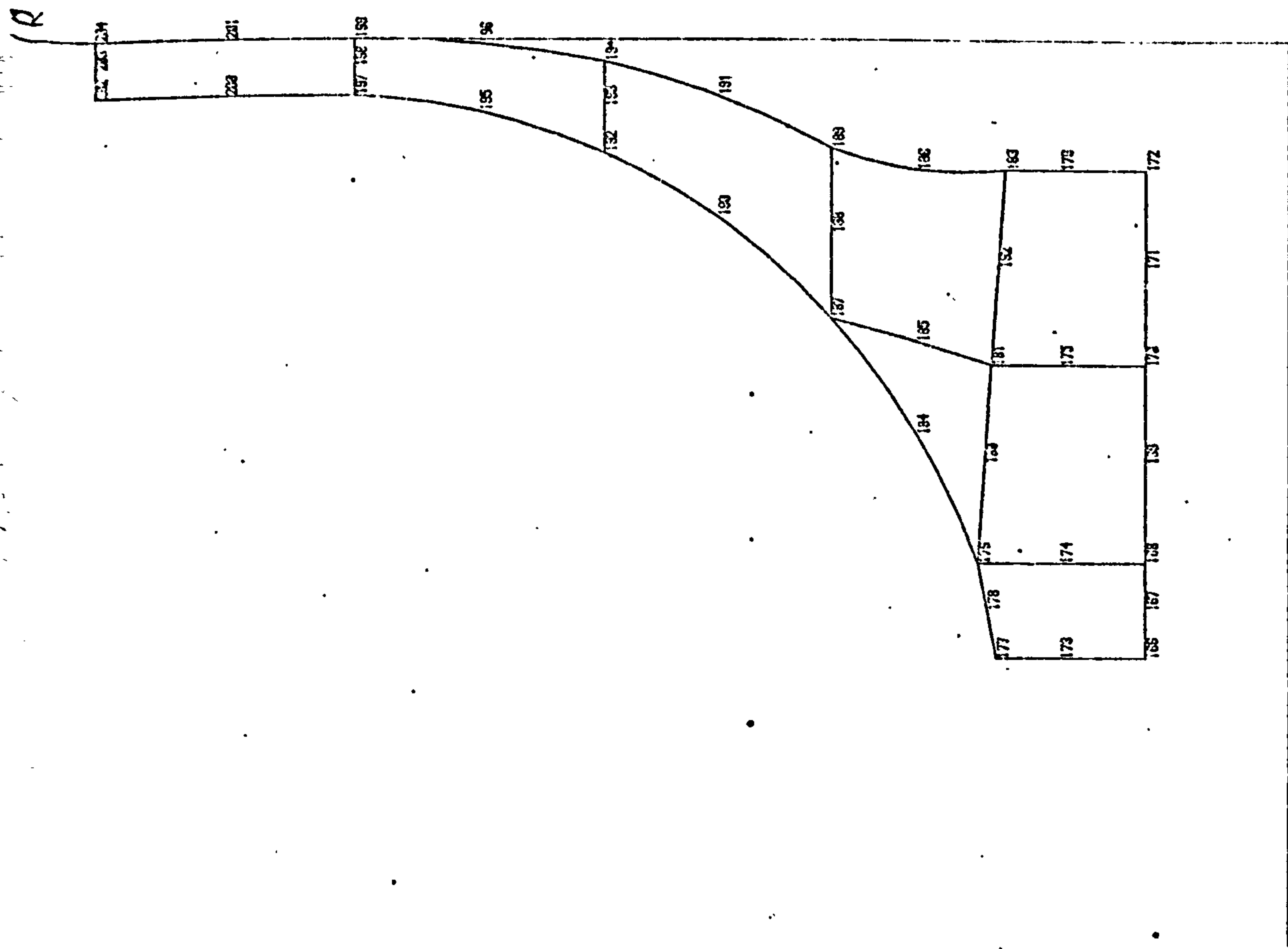


Z Y-Y Plane of Layer 3

DRAWN IN FIM SCALE: 55.80 TIMES THE SUPPLIED UNITS
Sheet 3

IMPELLER MODEL-P MESH-I

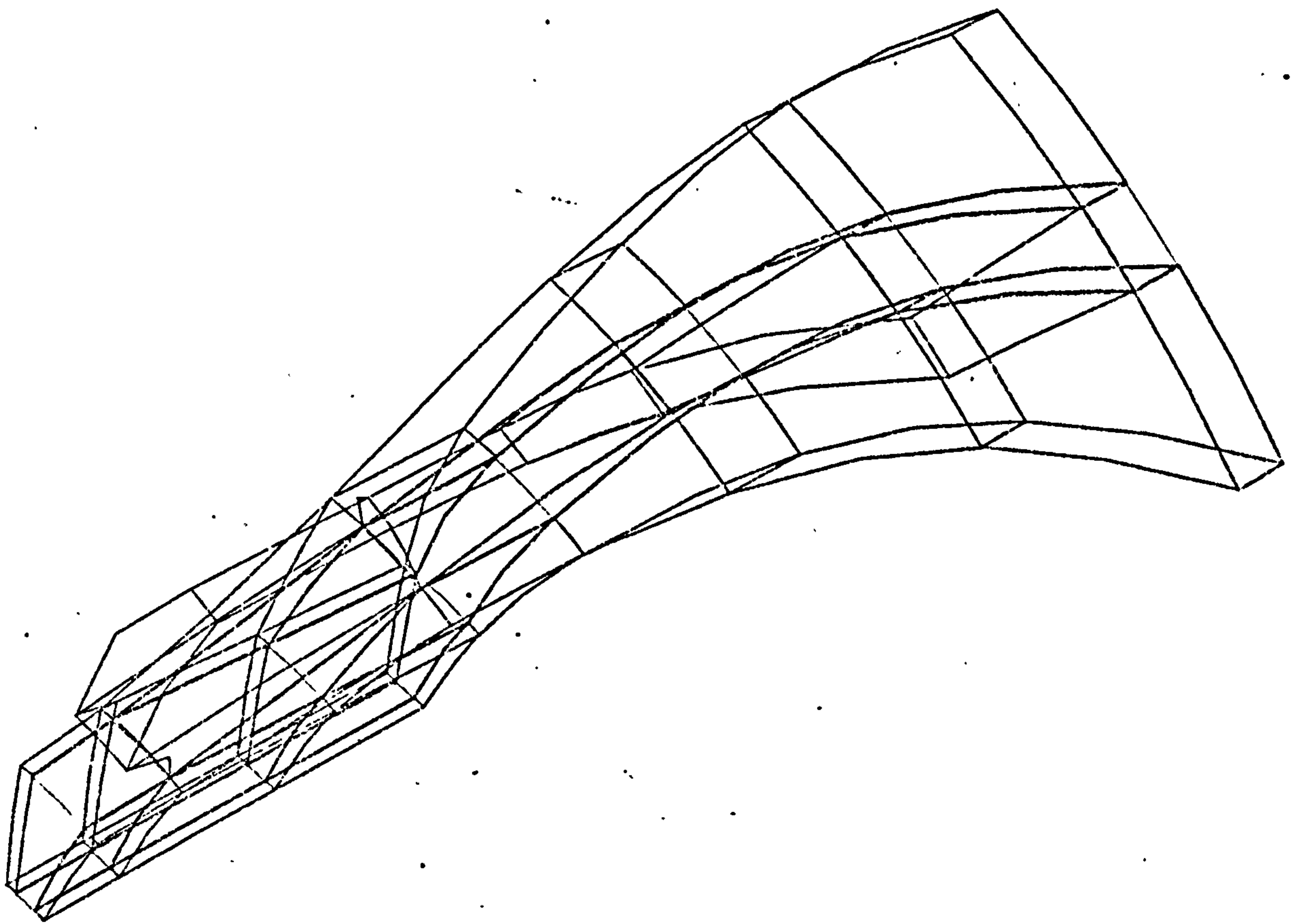
FIG 3.3 (e)



DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS
Sheet 4

IMPELLER MODEL-P MESH-I

FIG 3-3(f)



3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS

FIG 3-3(g) IMPELLER MODEL-P MESH-I

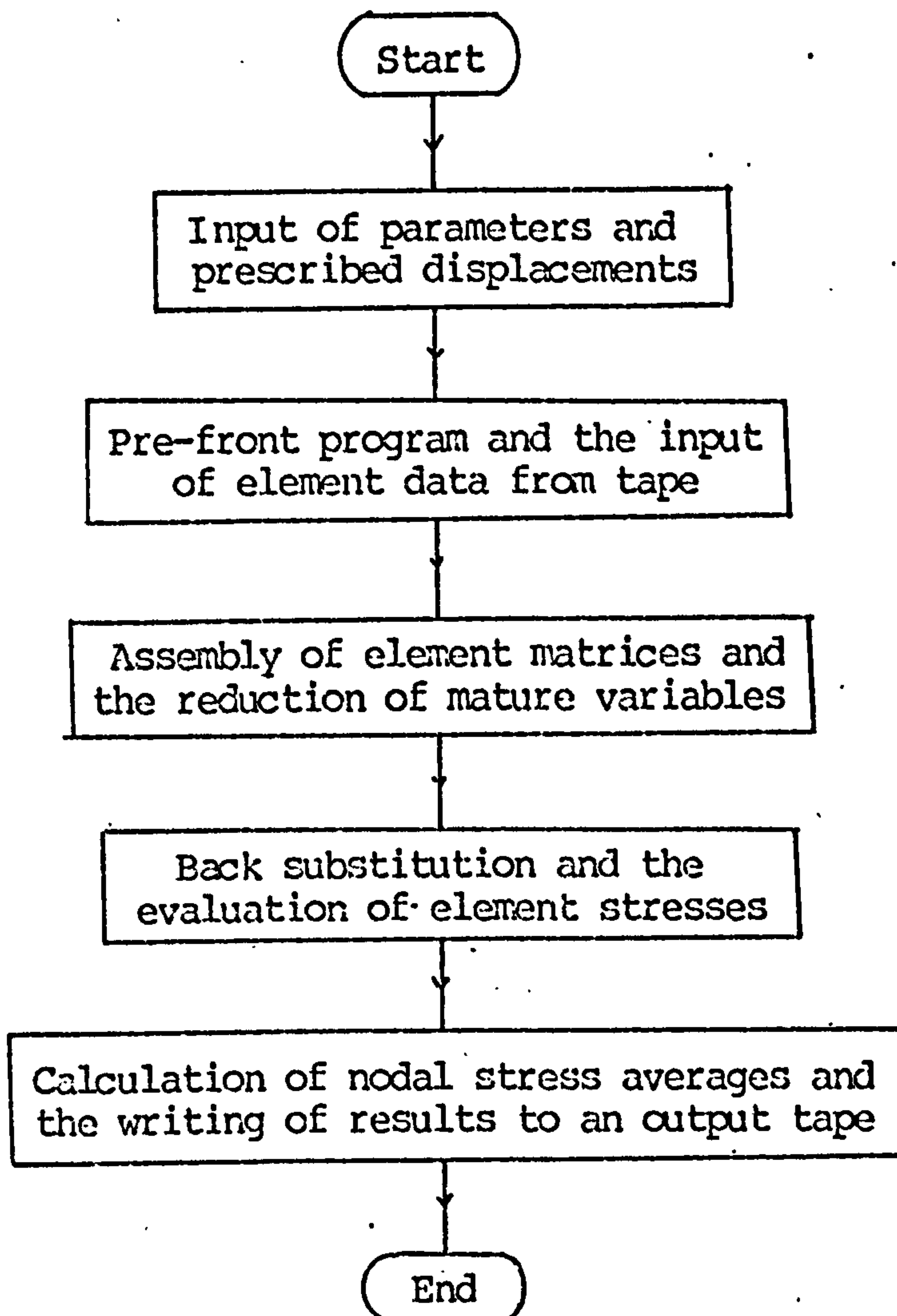


Fig 3.4 Flow diagram of Frontal Solution Program

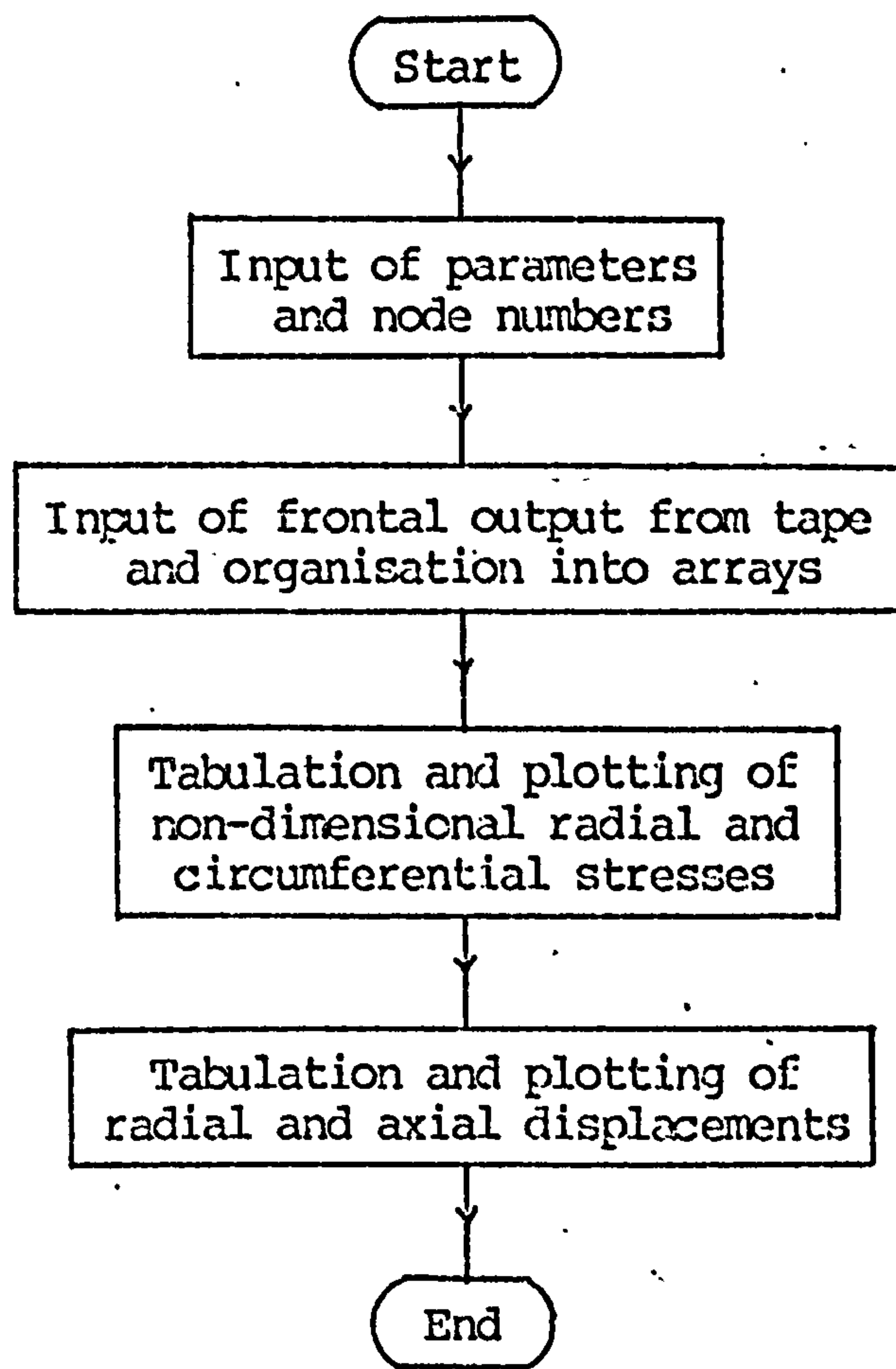


Fig 3.5 Flow diagram of Output Processor Program

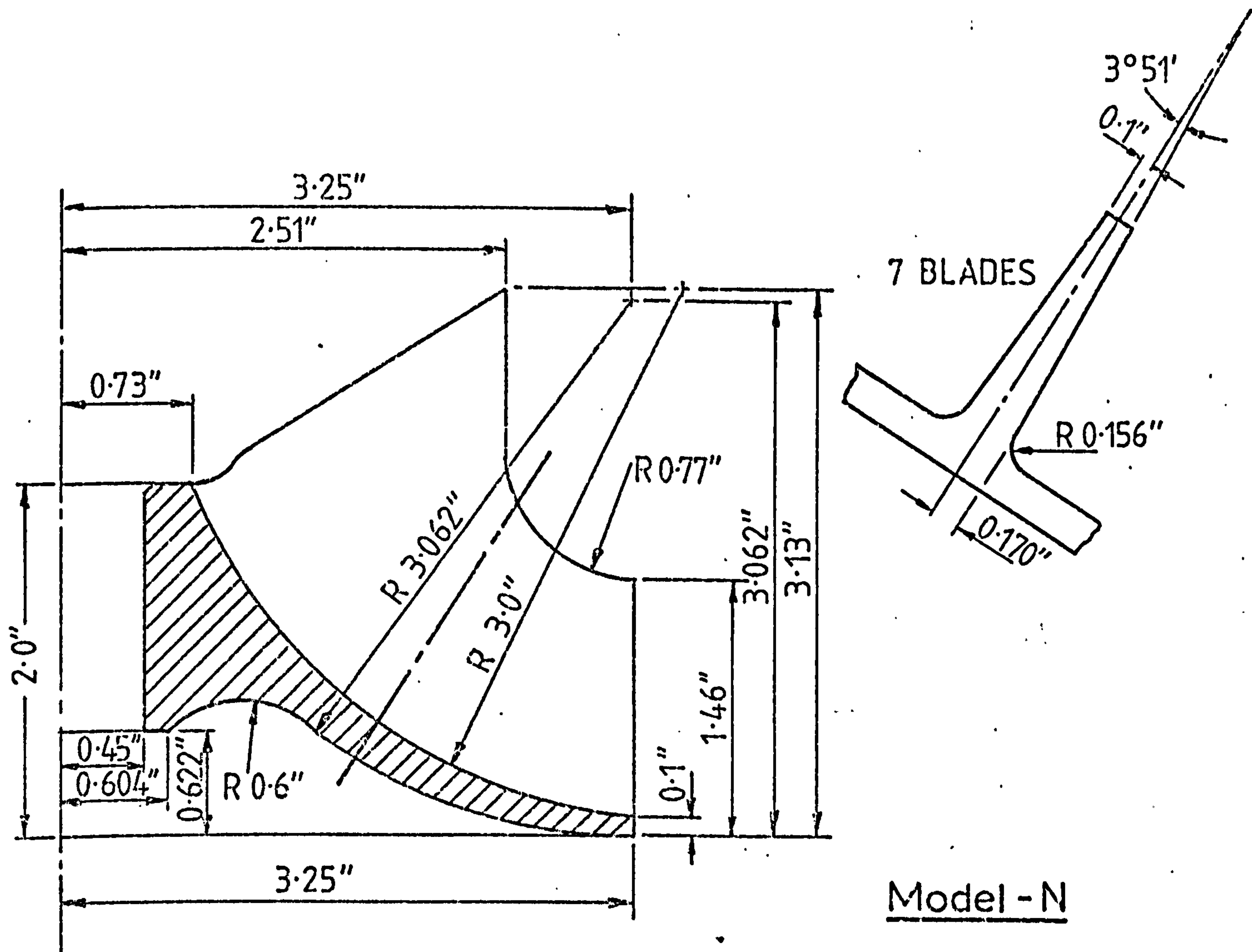
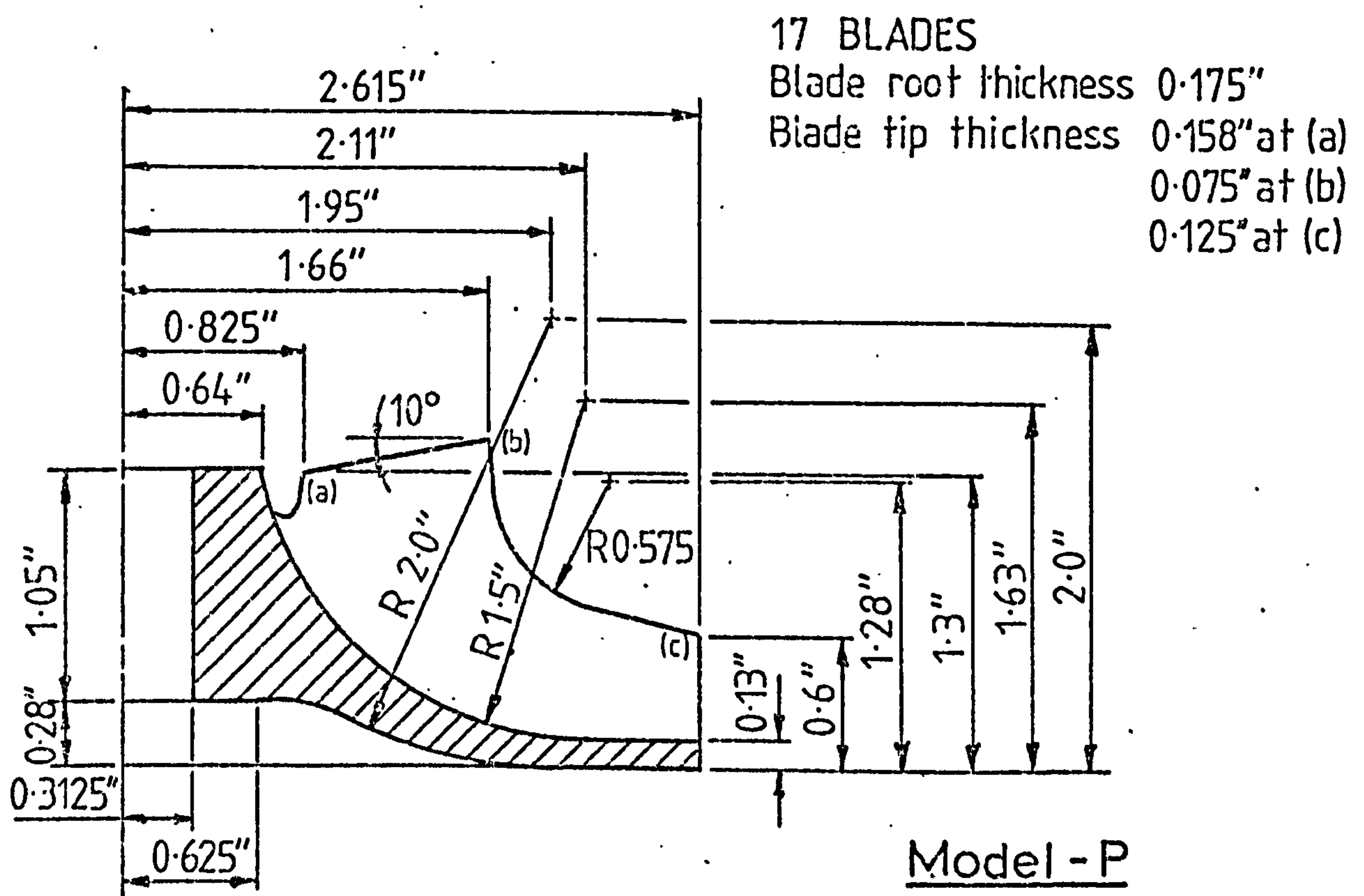
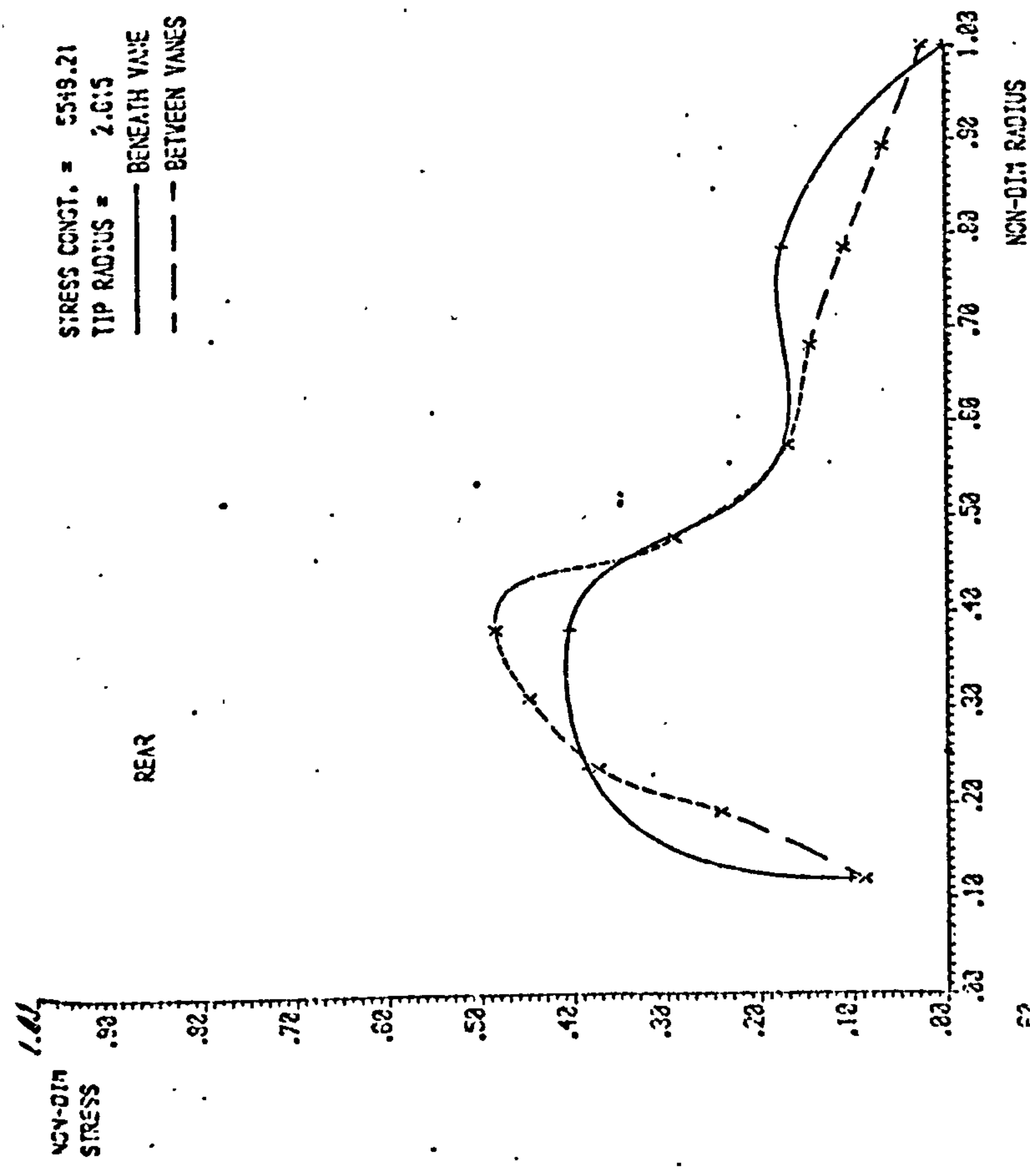
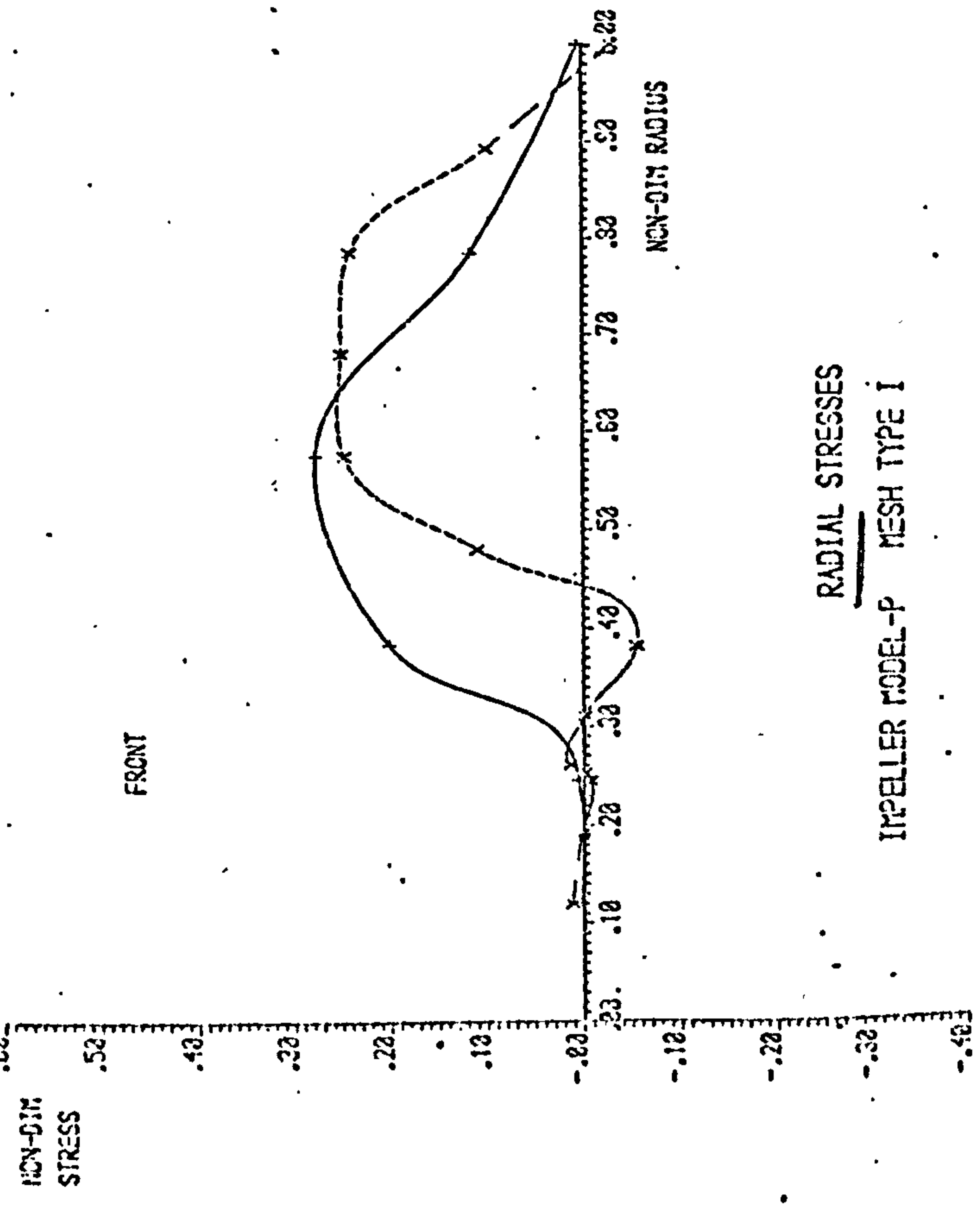
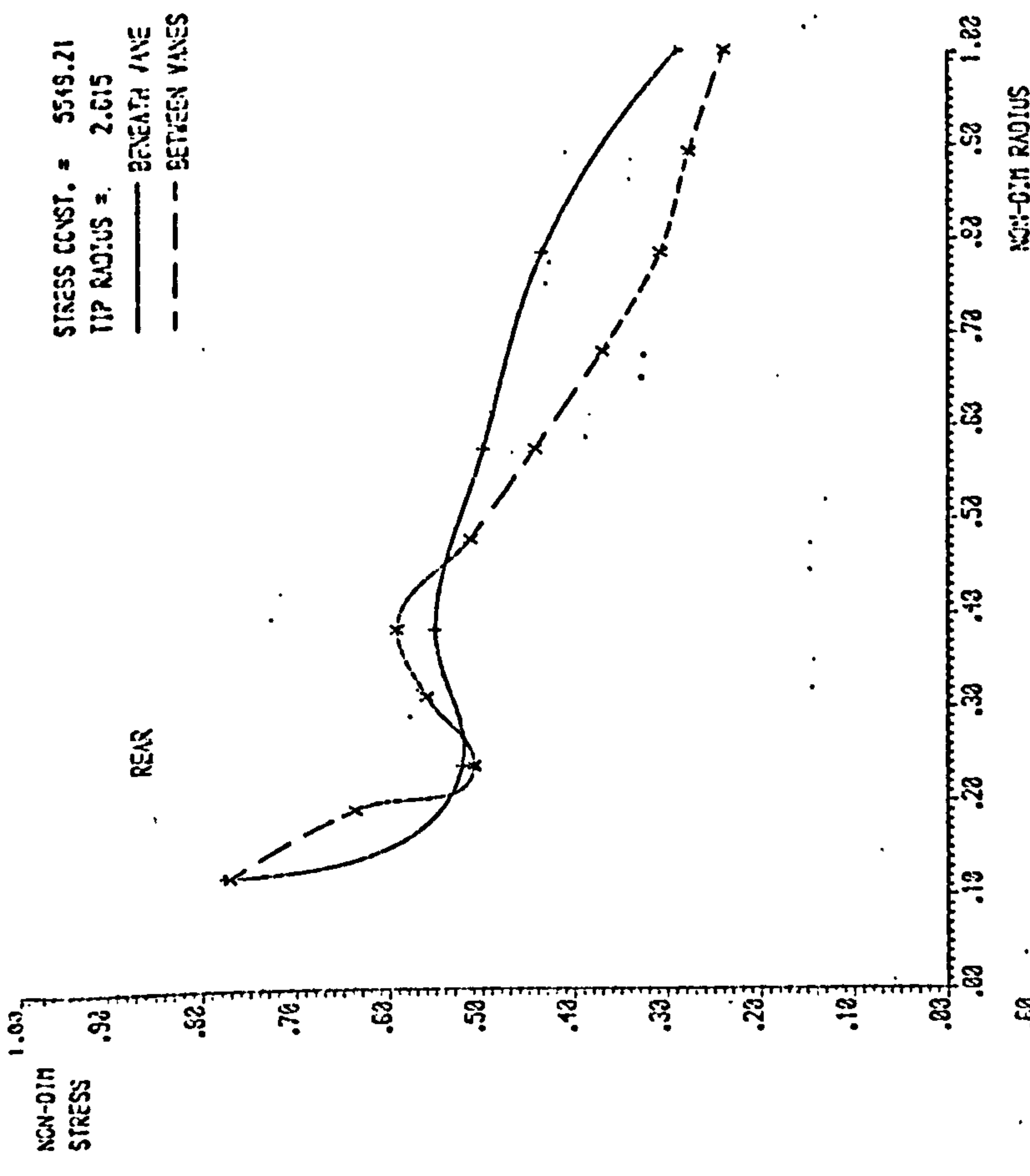
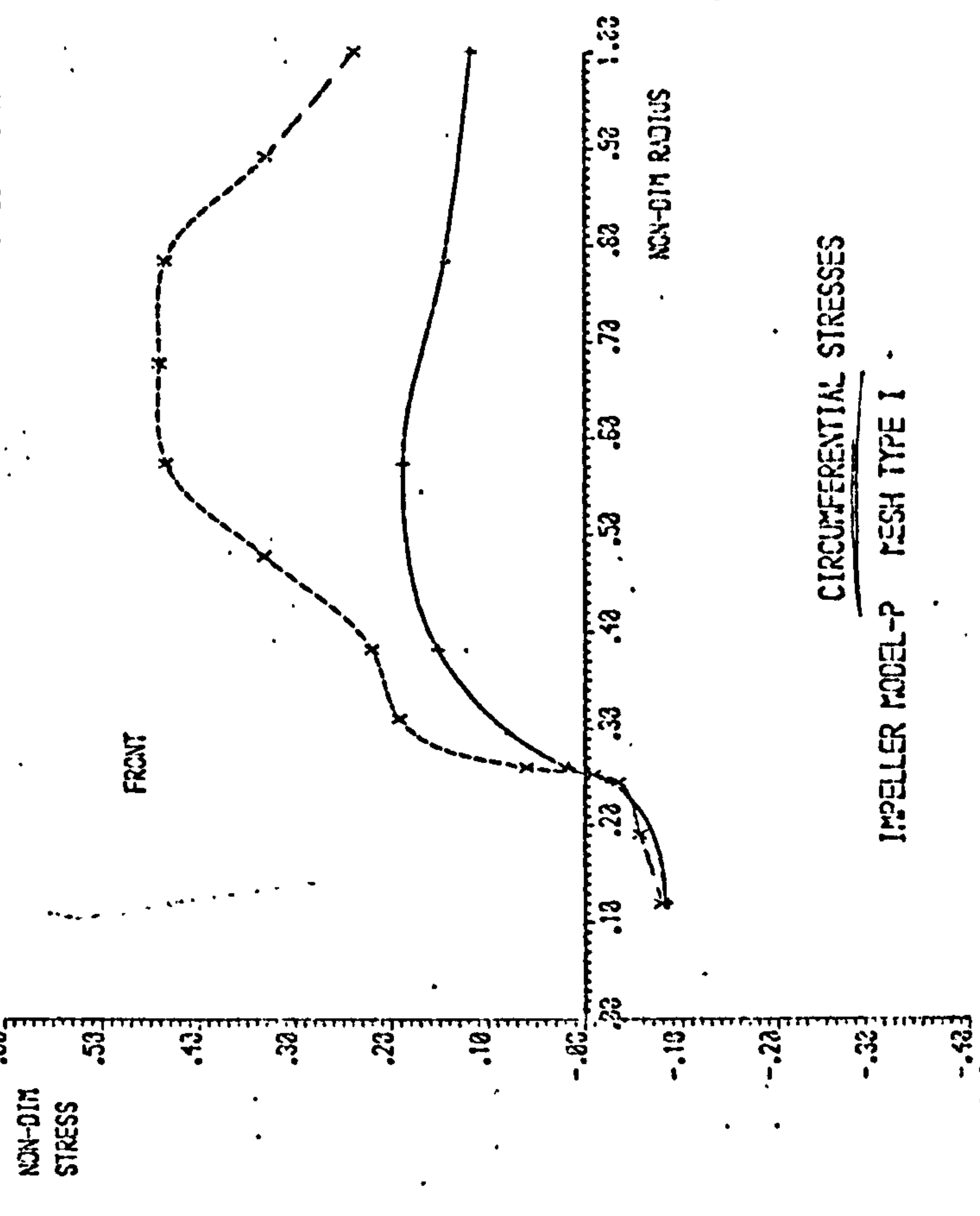


FIG 4-1 Hawker Siddeley Dynamics Limited
Impeller Models.



RADIAL STRESSES
IMPELLER MODEL-P MESH TYPE I



CIRCUMFERENTIAL STRESSES
IMPELLER MODEL-P MESH TYPE I

FIG 4-2(a)

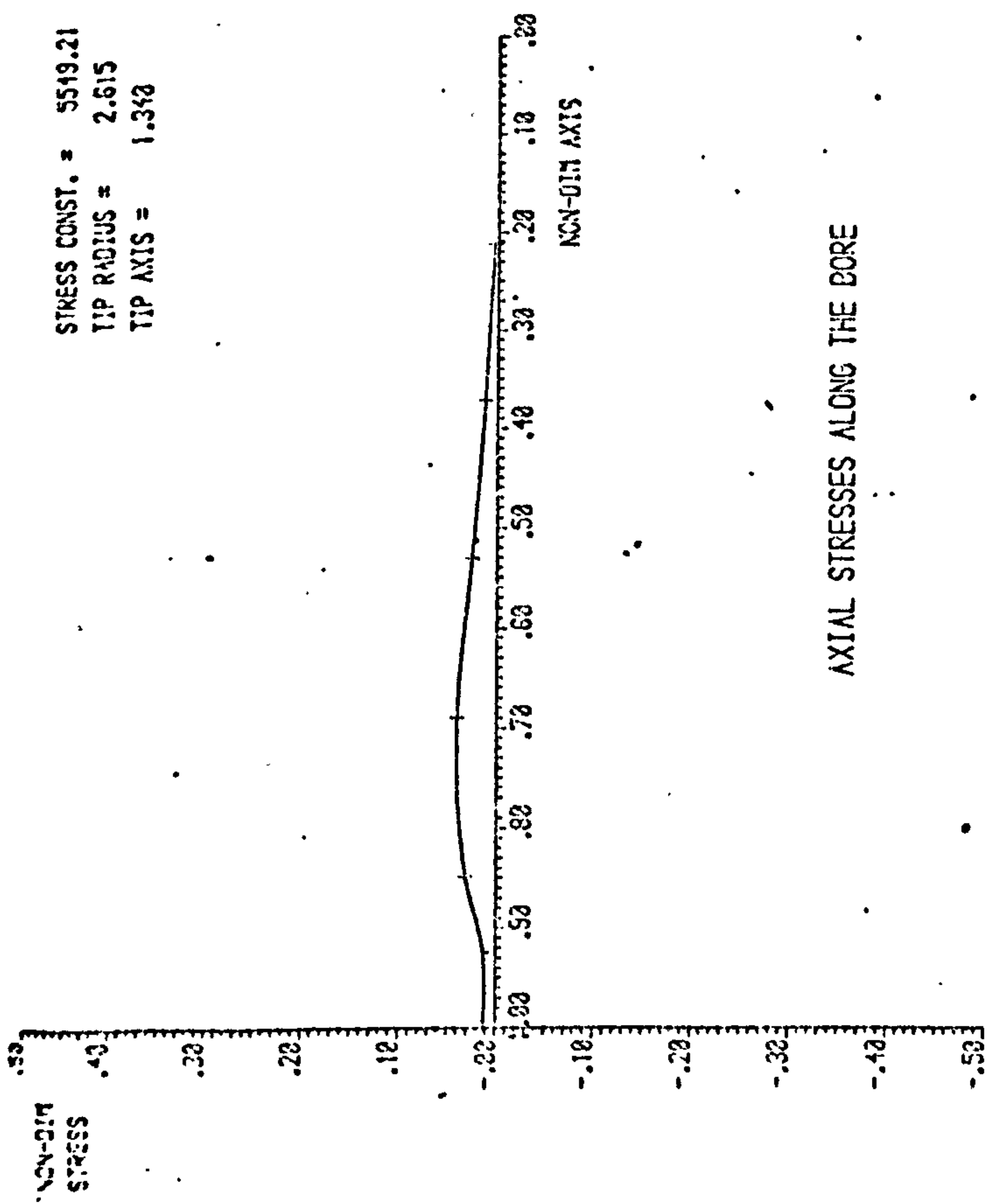
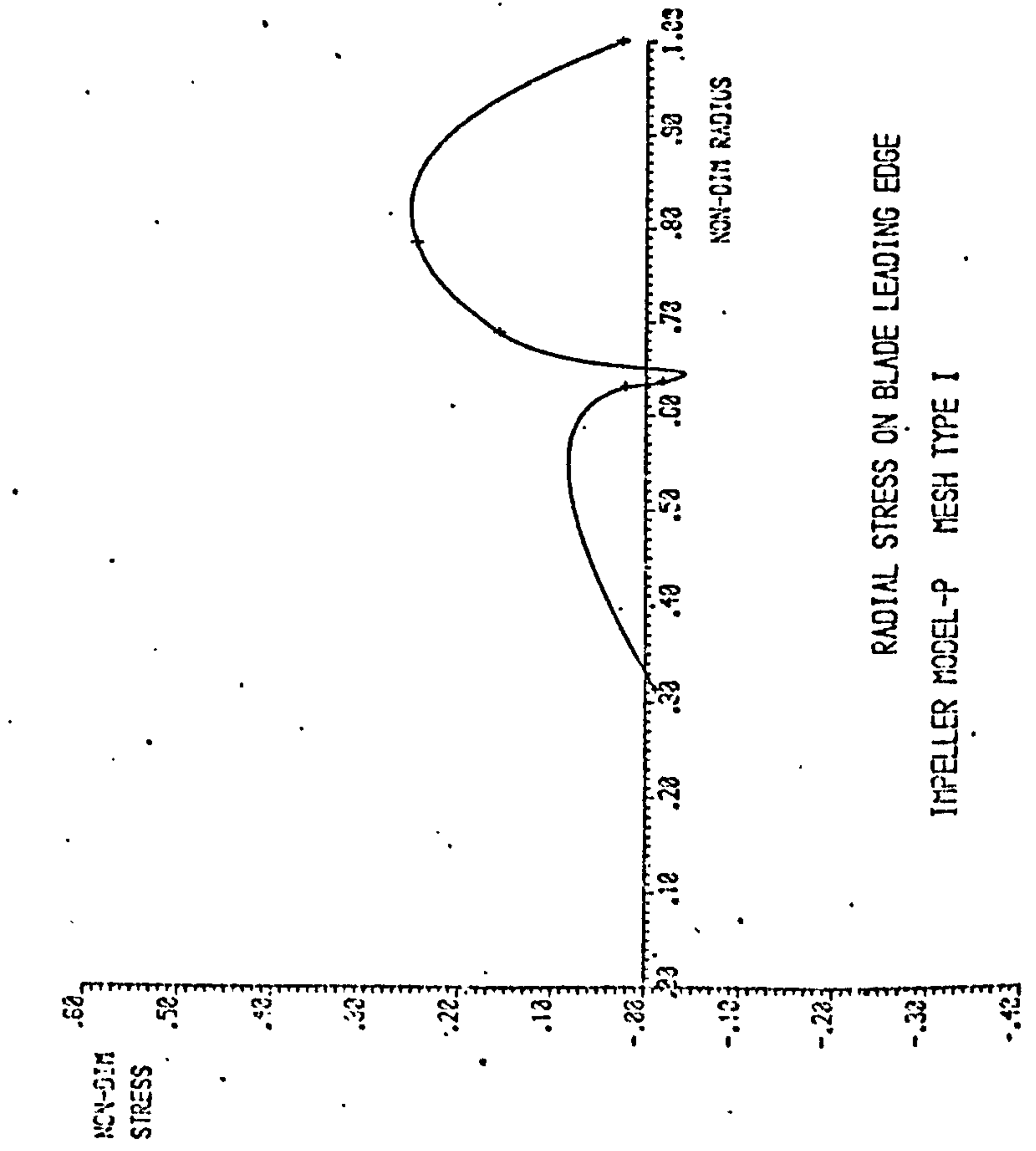
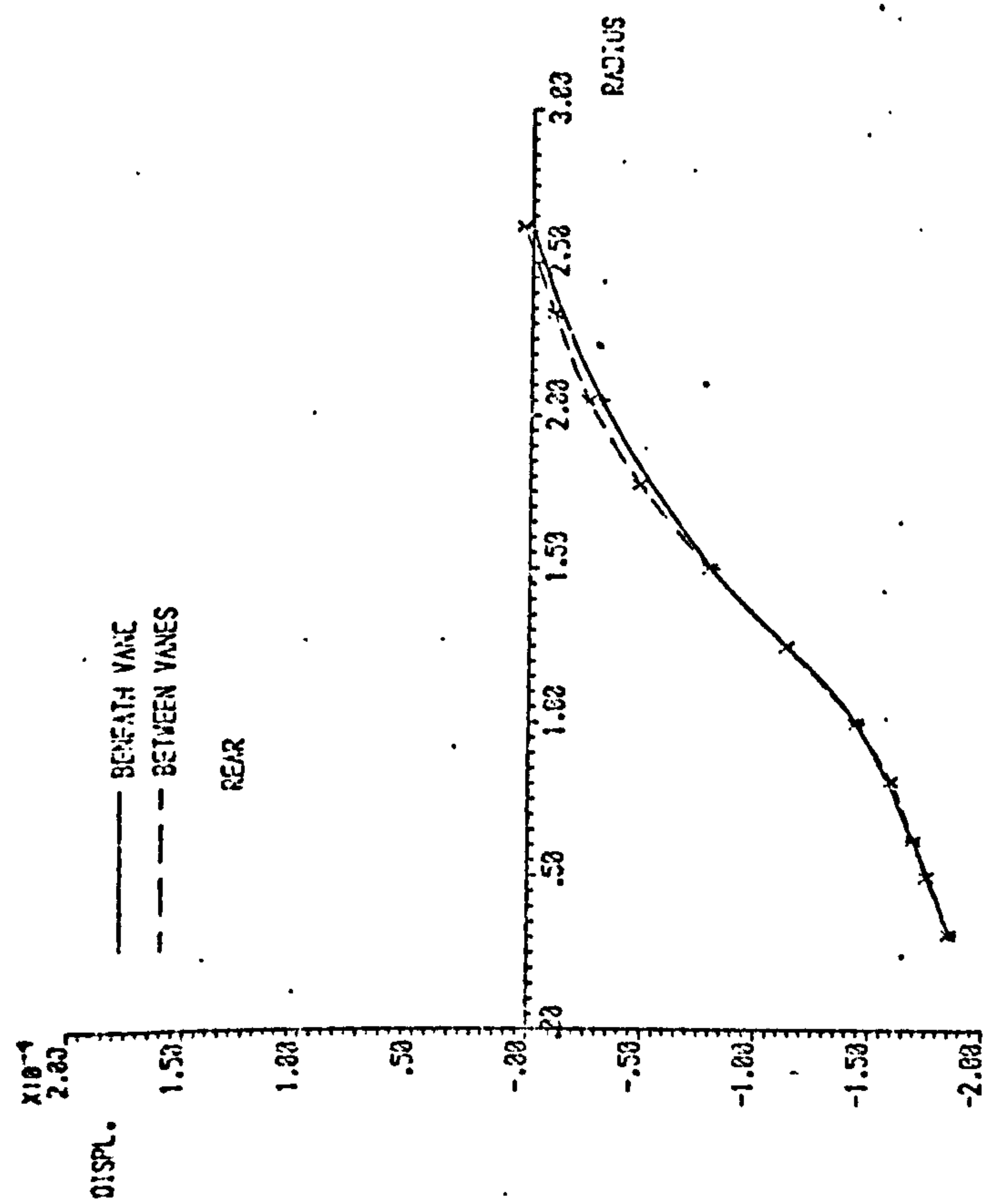
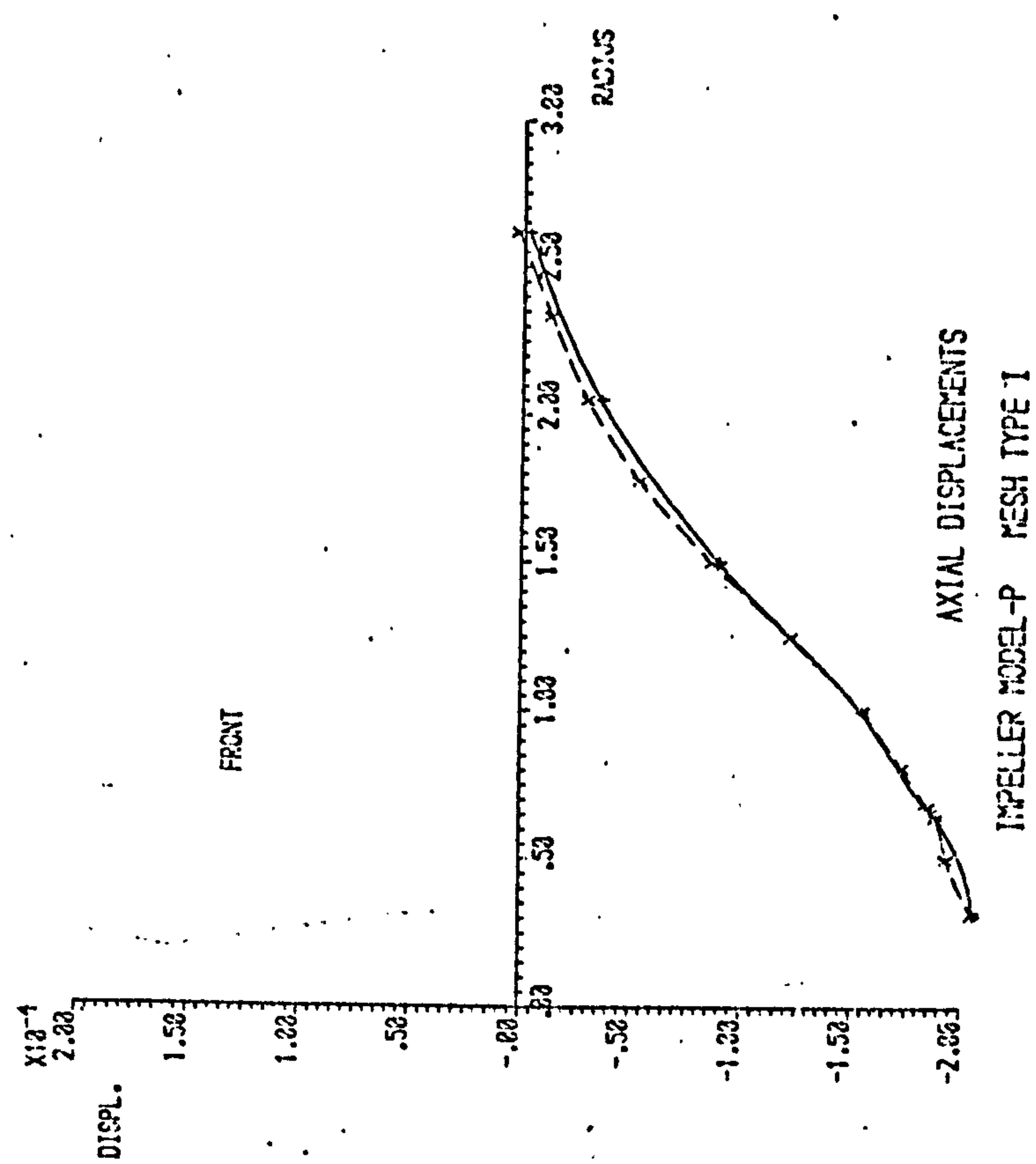
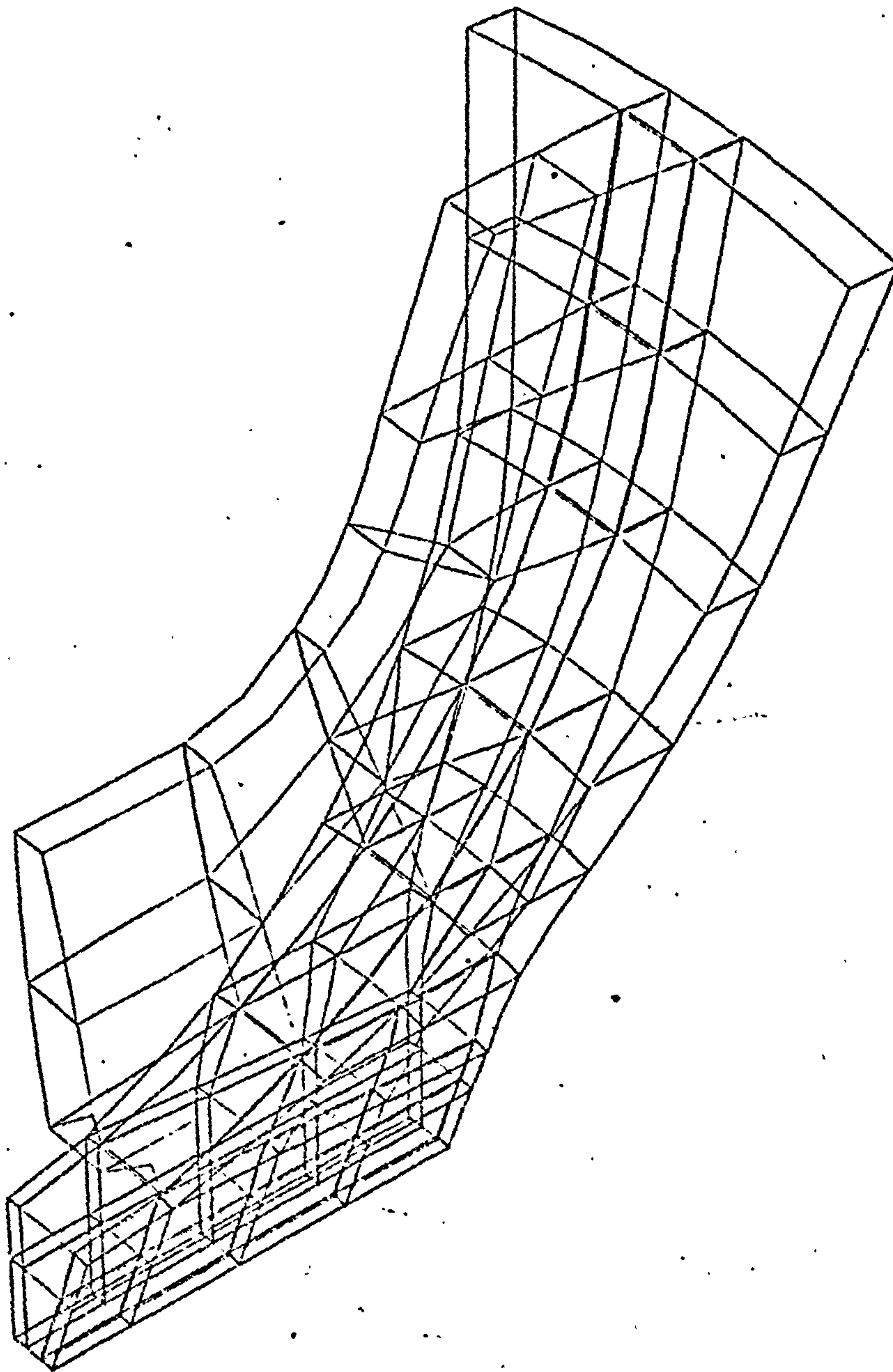
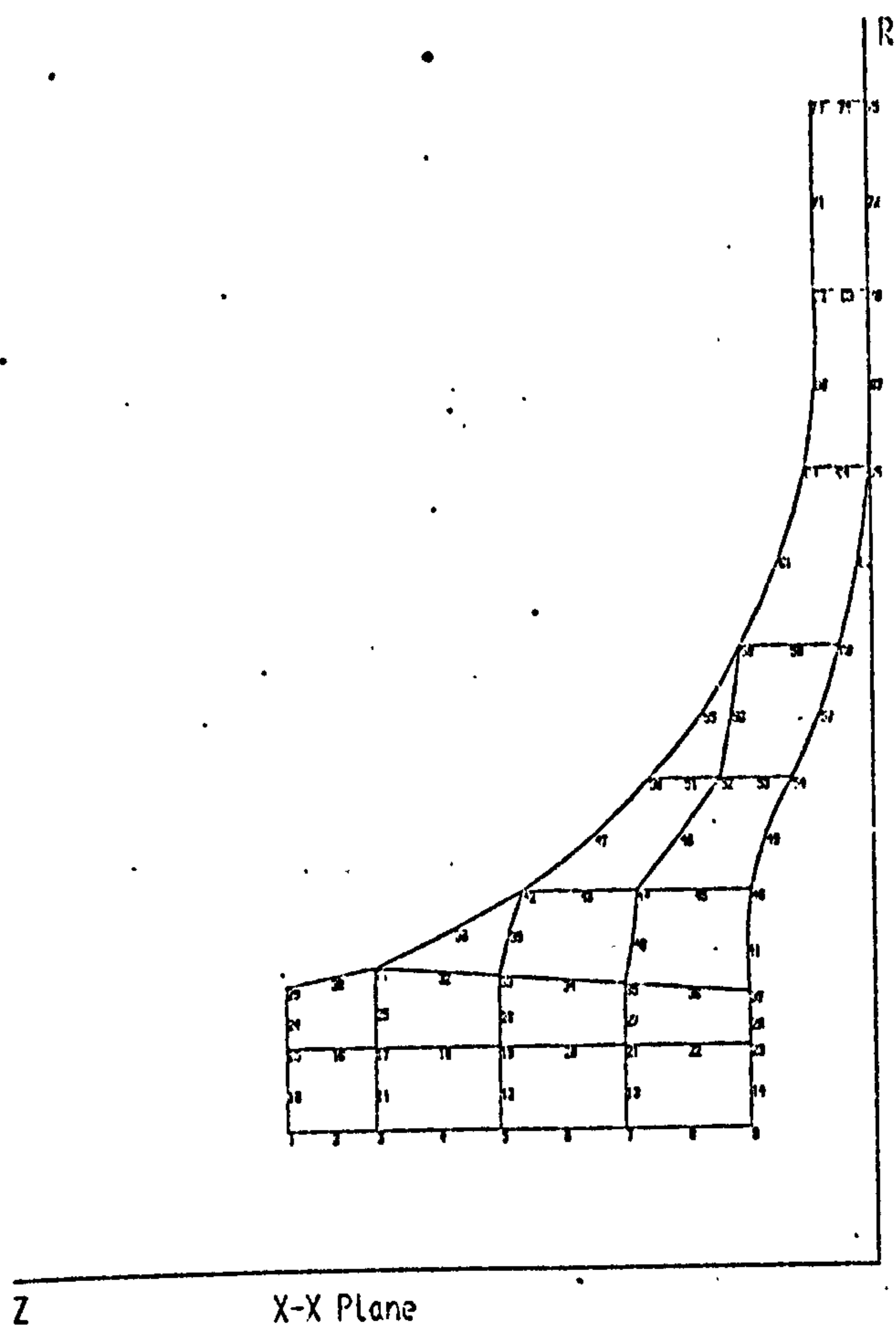


FIG 4-2(b)

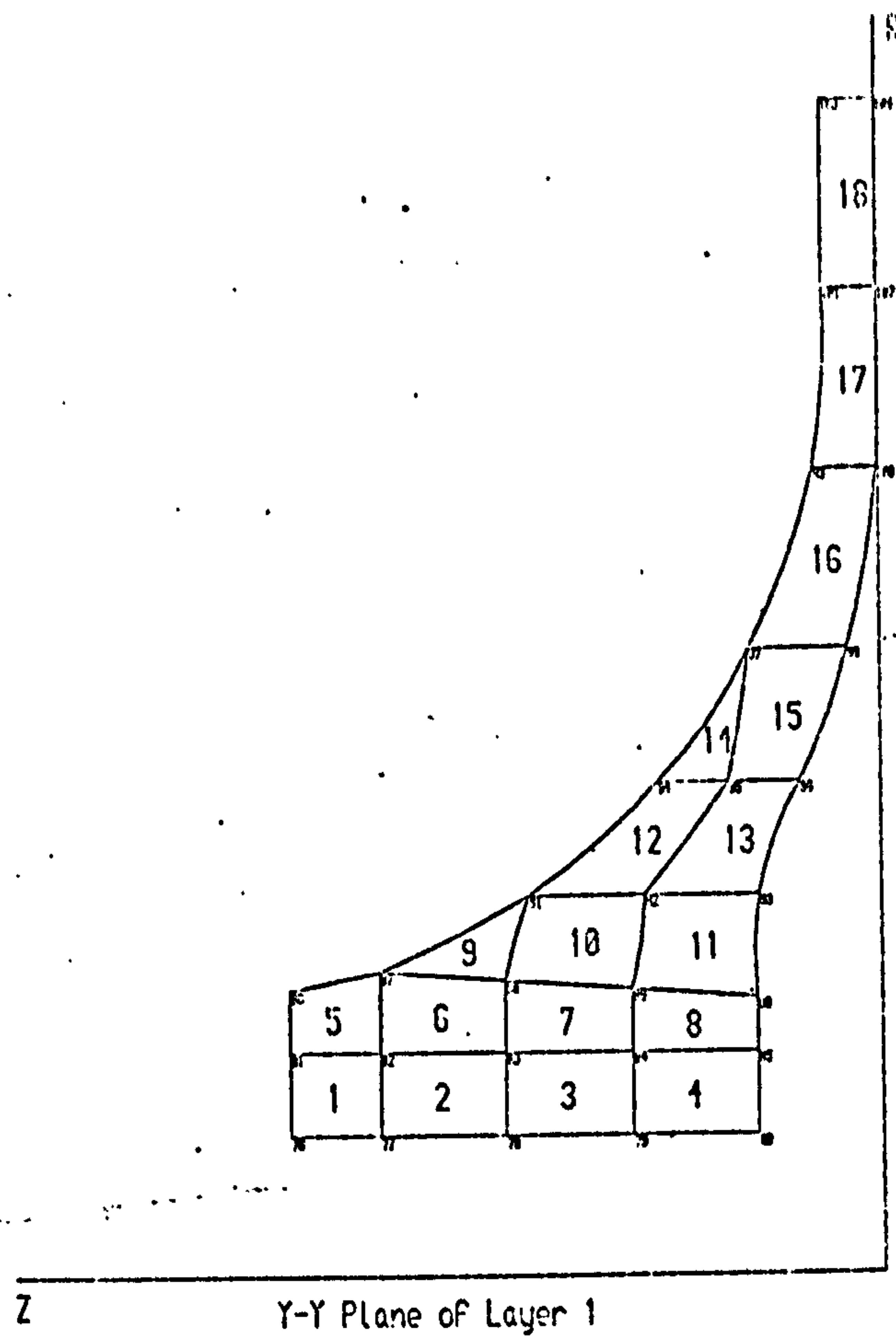


3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS

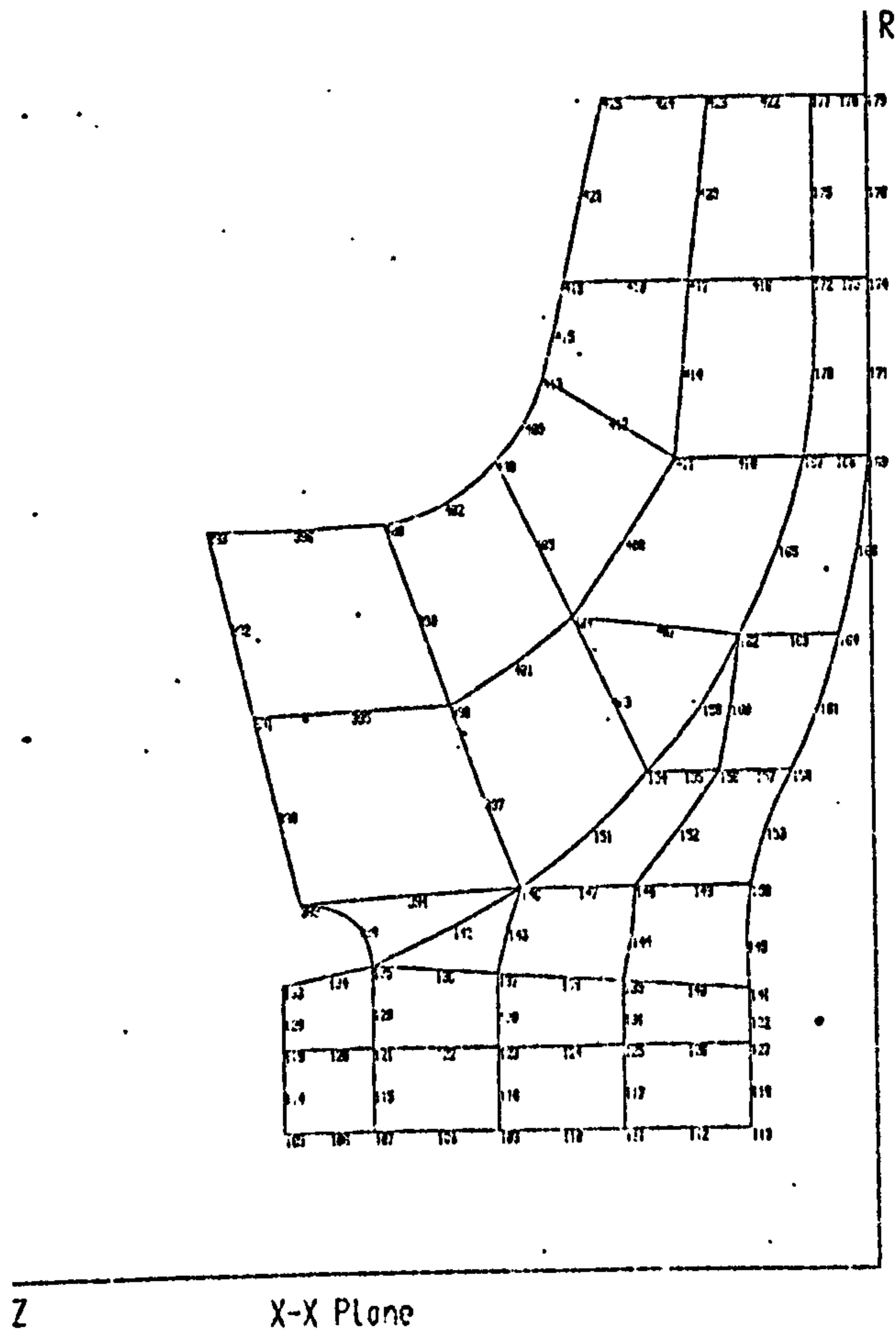
FIG 4.3(a) IMPELLER MODEL-P MESH TYPE II



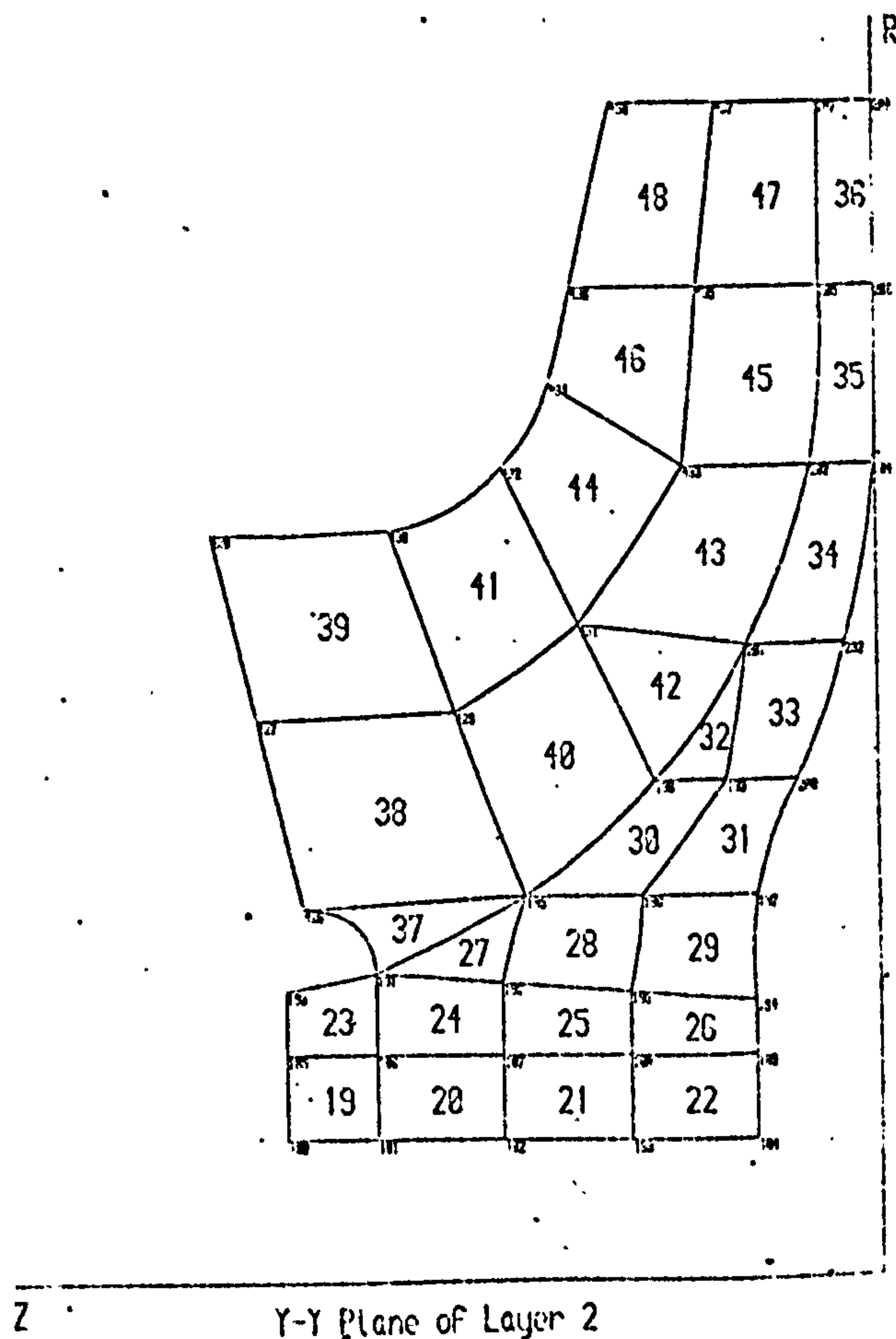
IMPELLER MODEL-P MESH TYPE II



DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS
Sheet 1



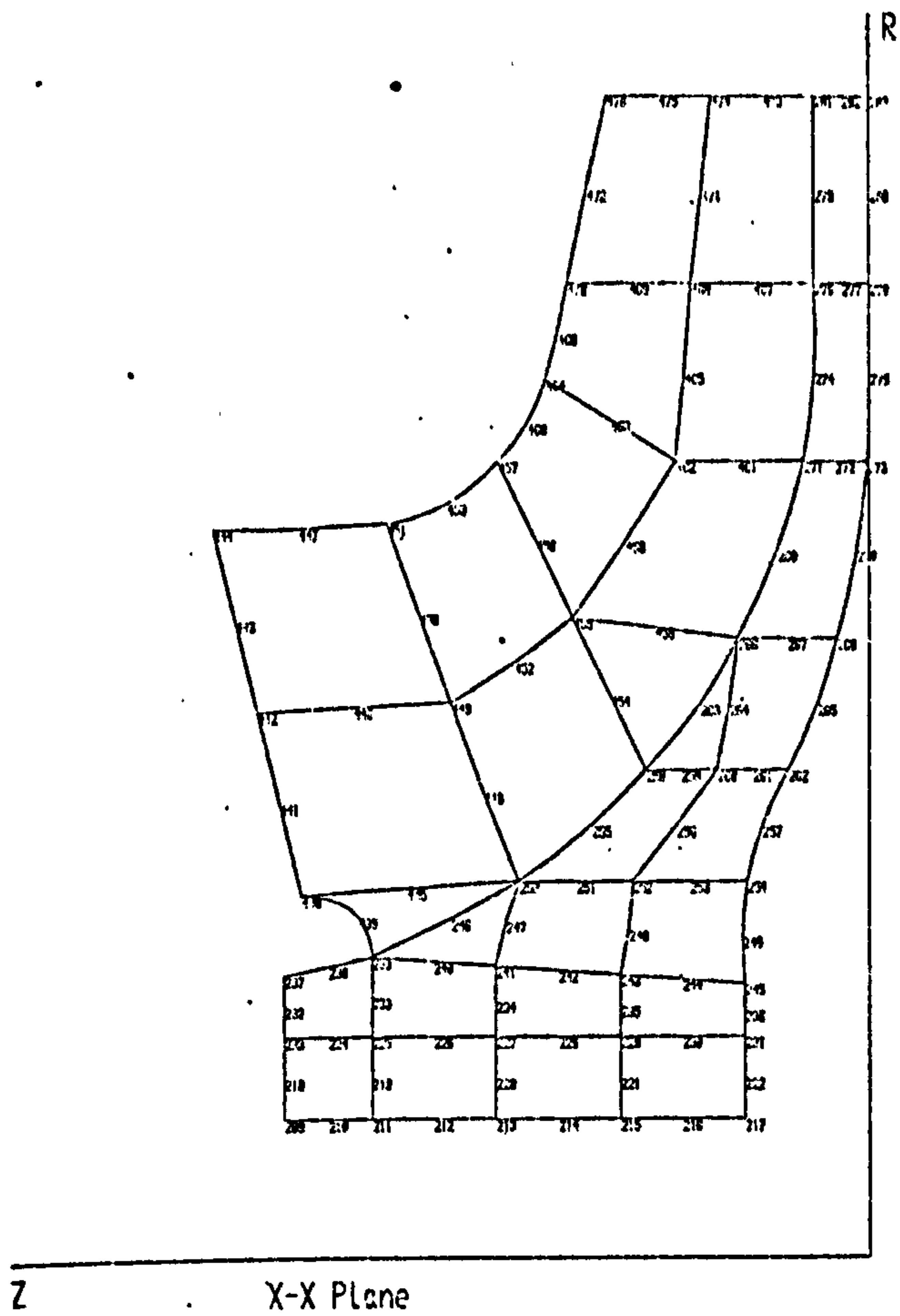
IMPELLER MODEL-P MESH TYPE II



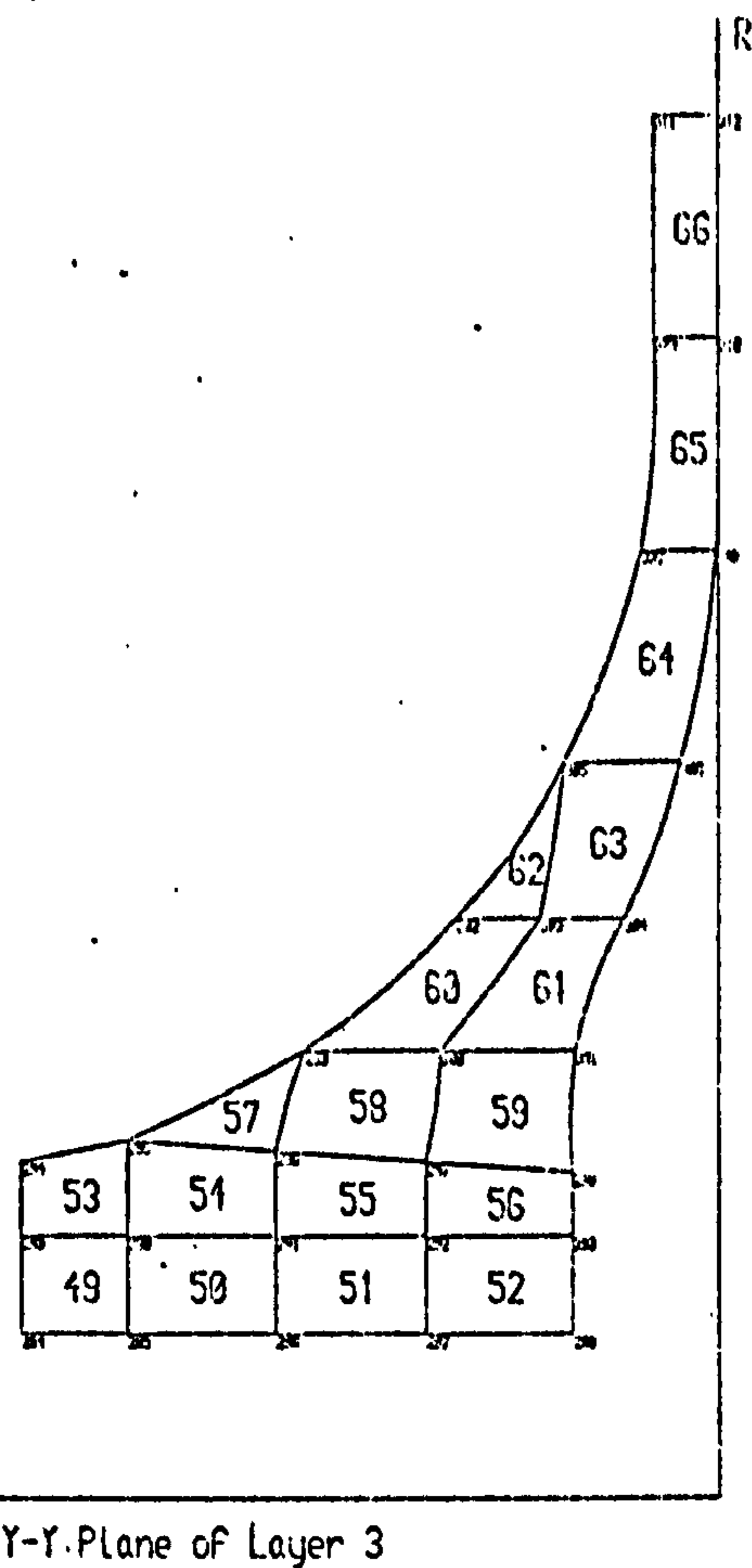
DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS
Sheet 2

FIG 4.3 (b)

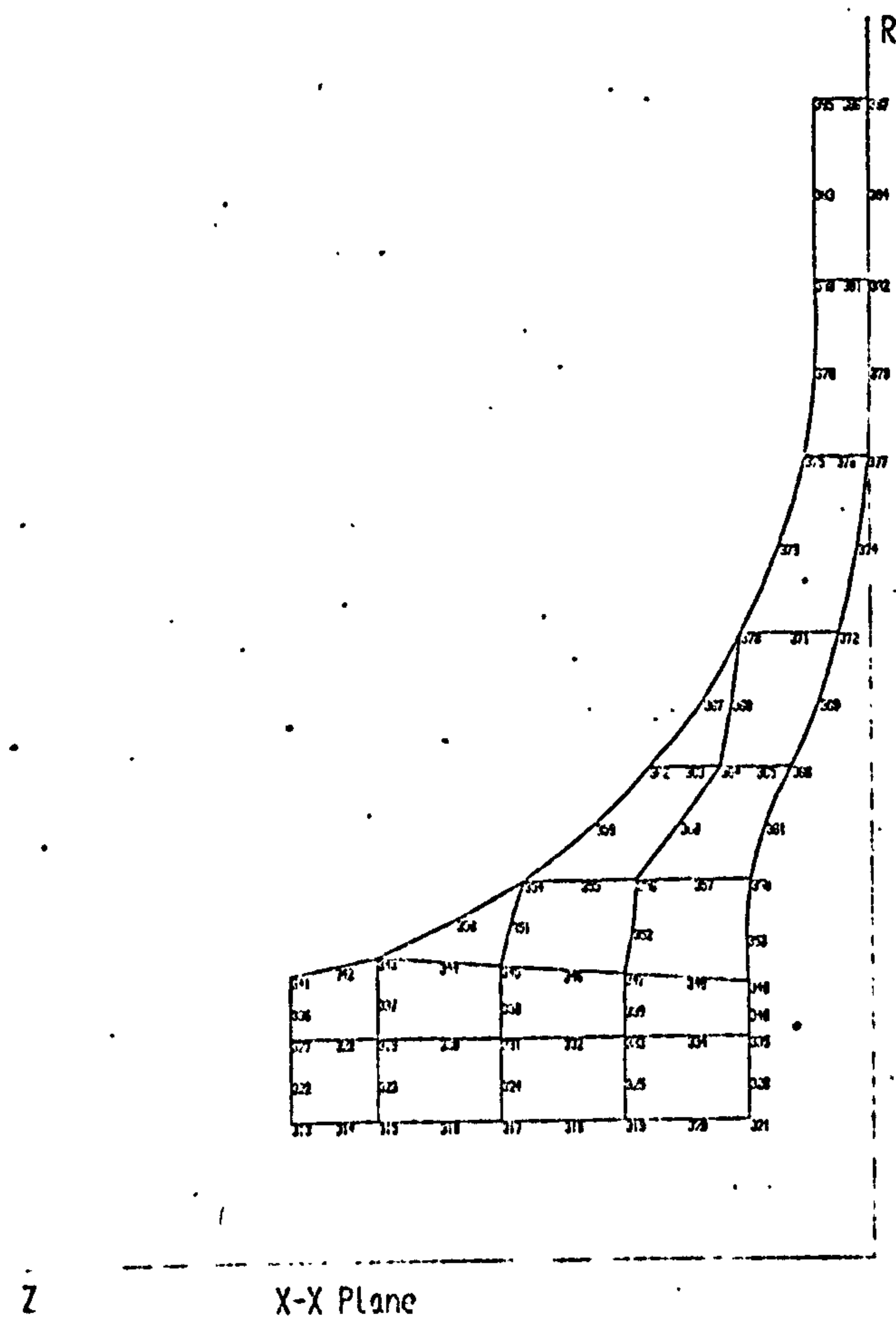
121.



IMPELLER MODEL-P MESH TYPE II



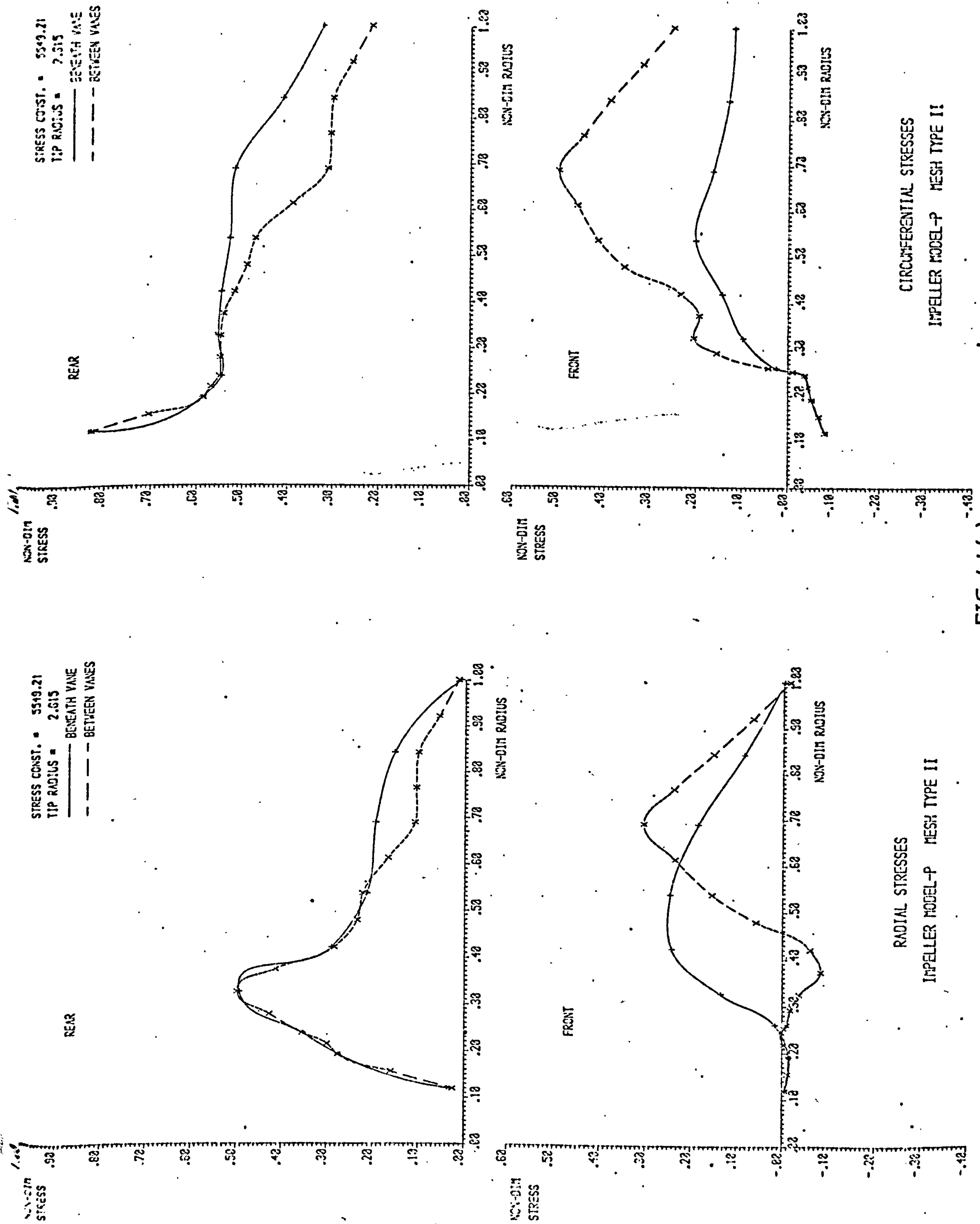
DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS
Sheet 3



IMPELLER MODEL-P MESH TYPE II

DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS
Sheet 4

FIG 4-3(c)



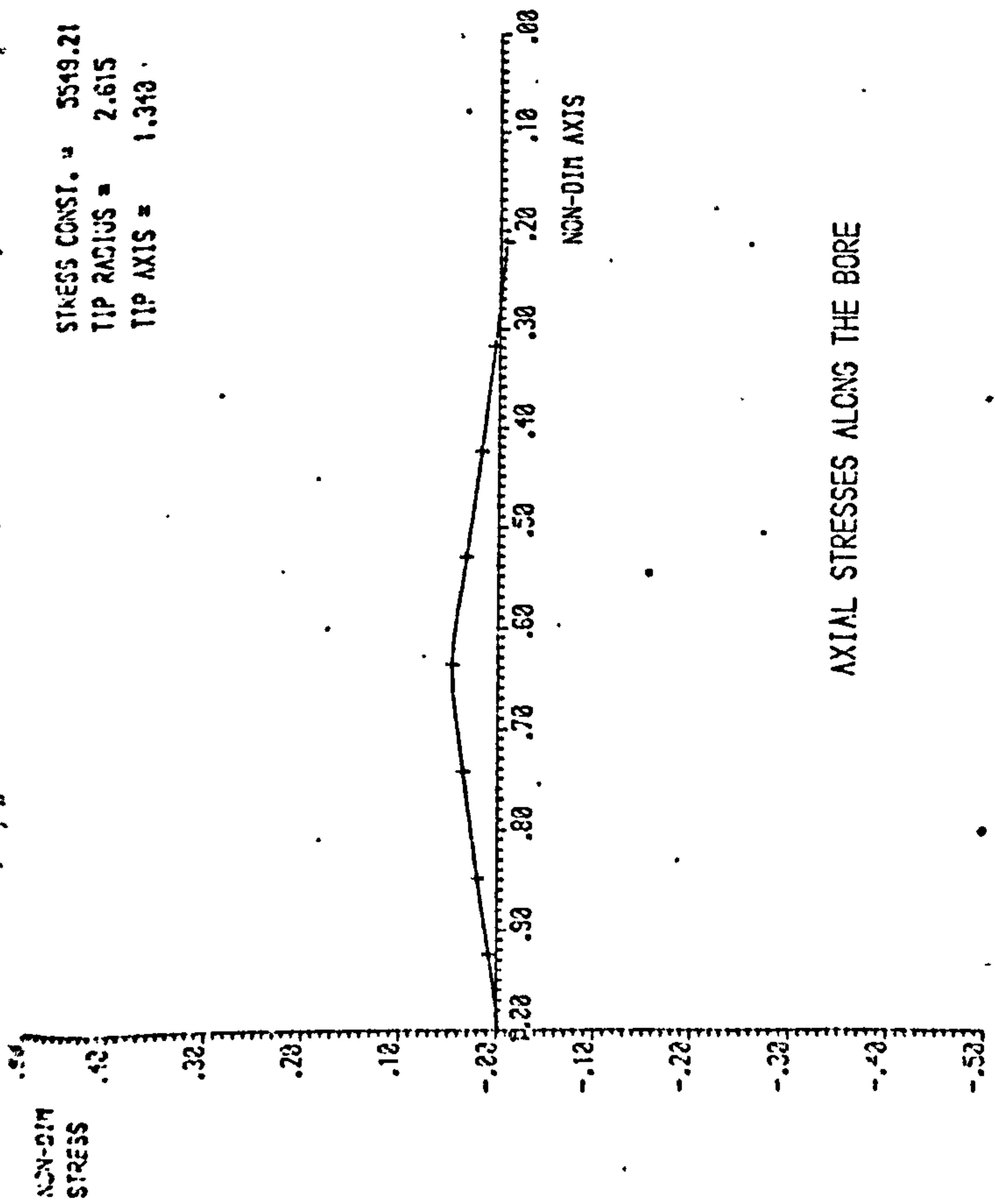
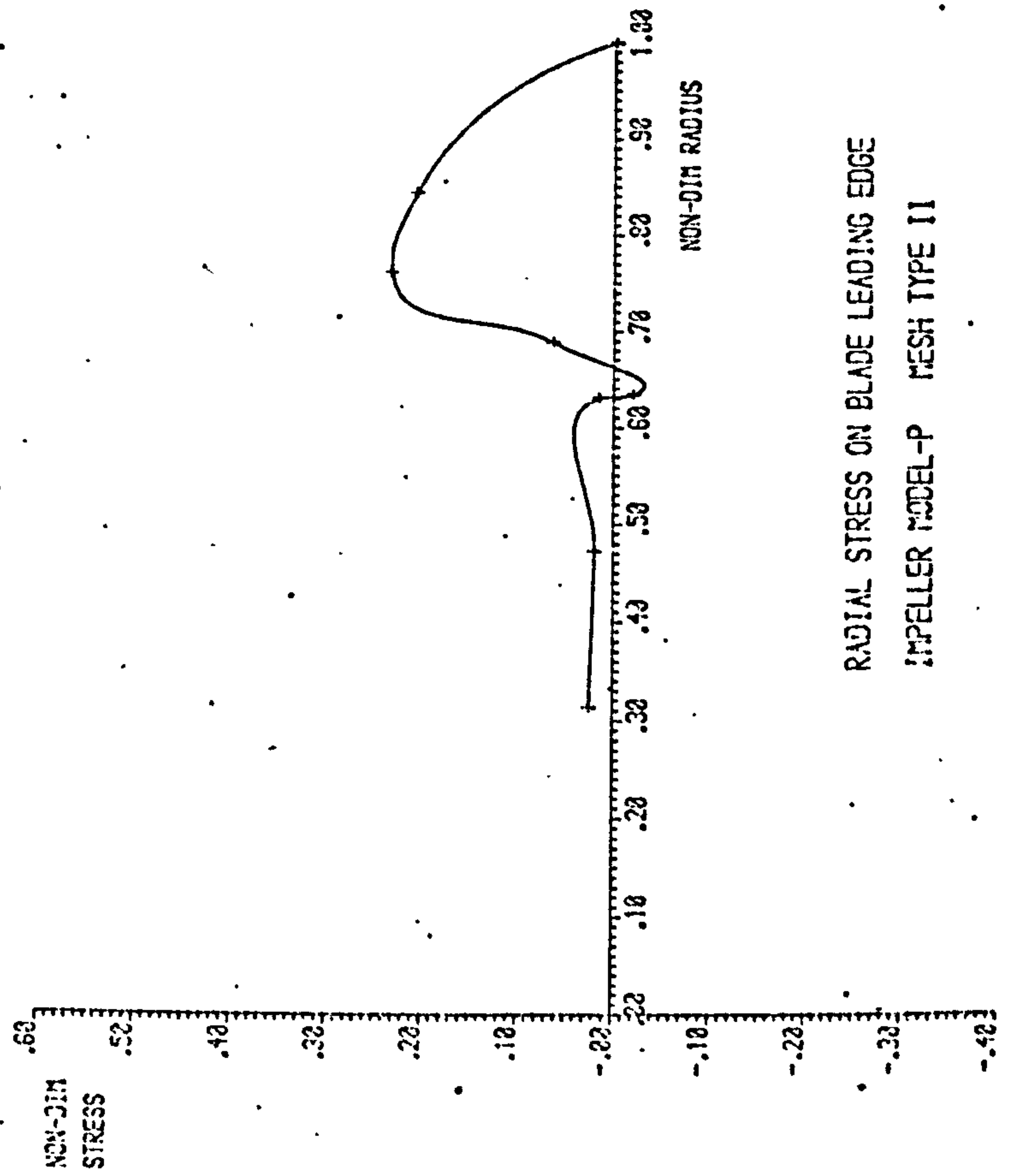
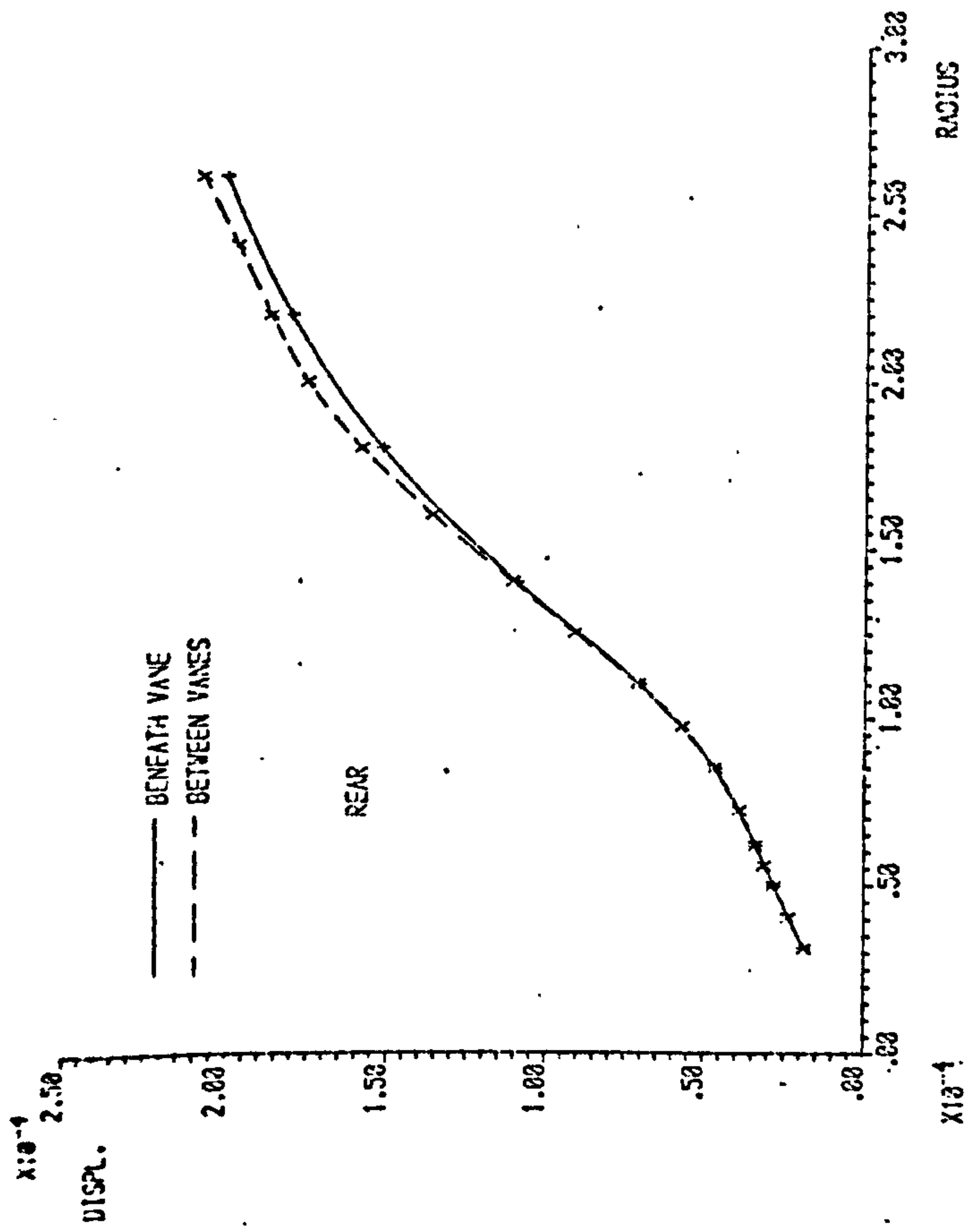
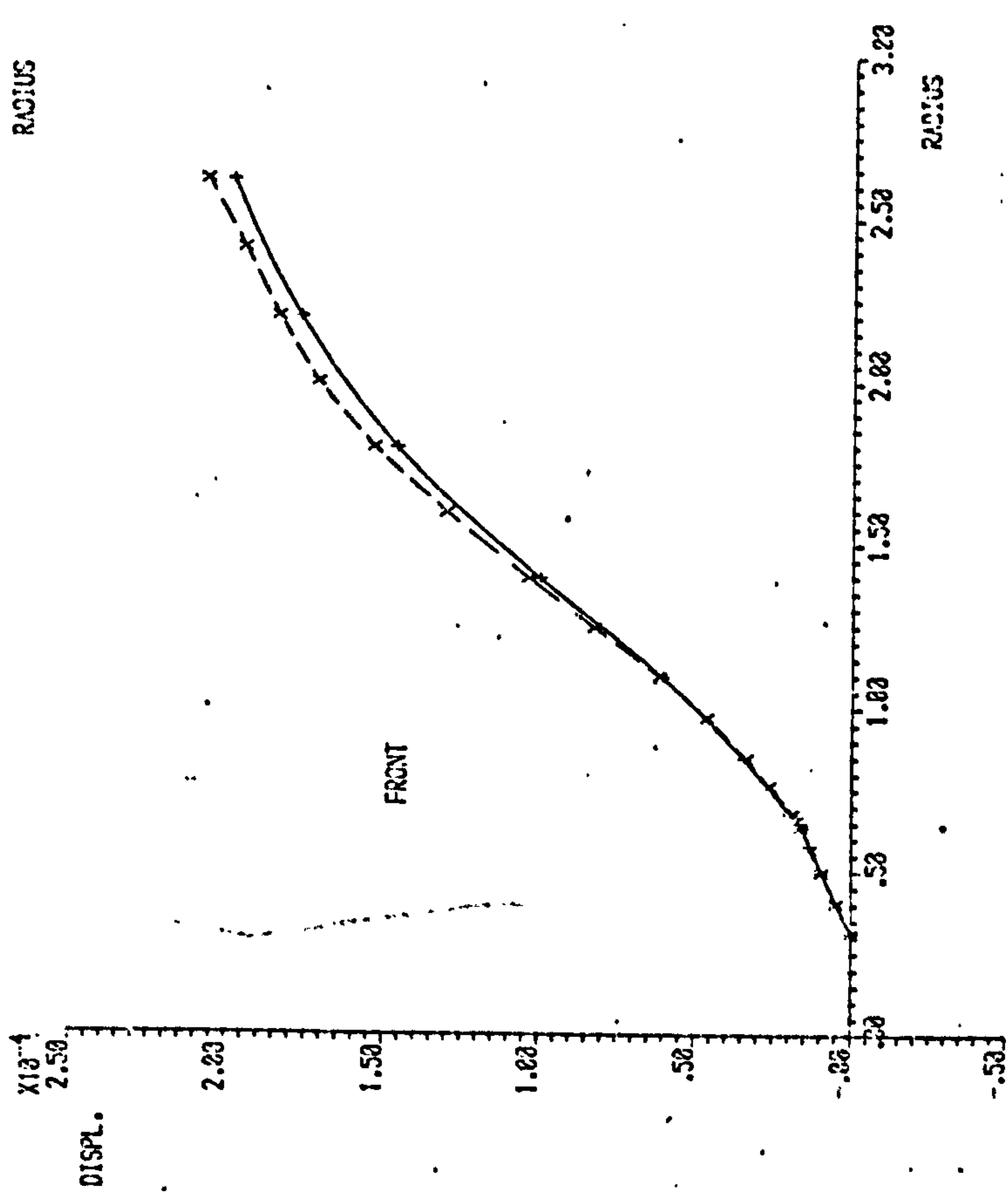
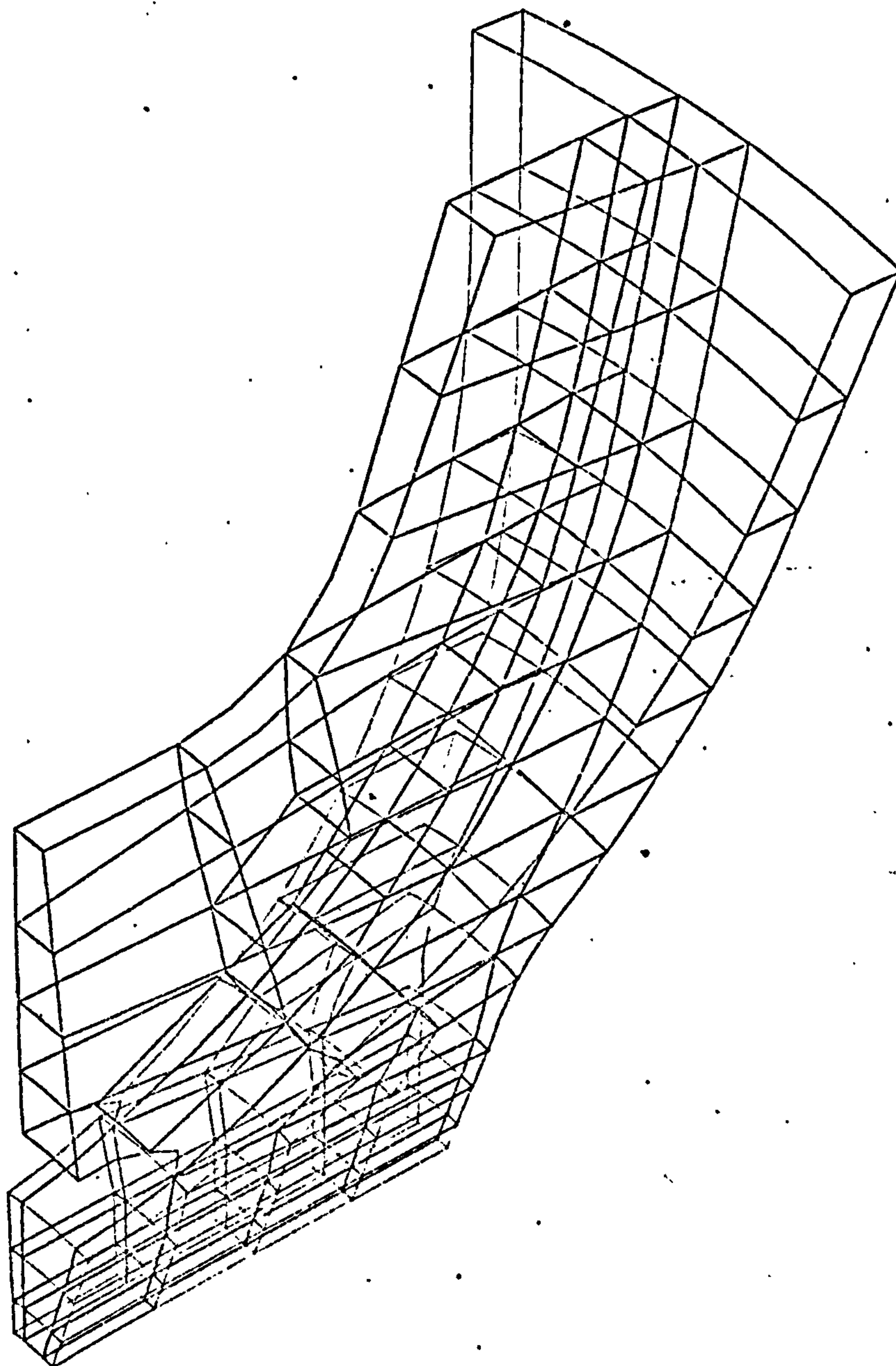
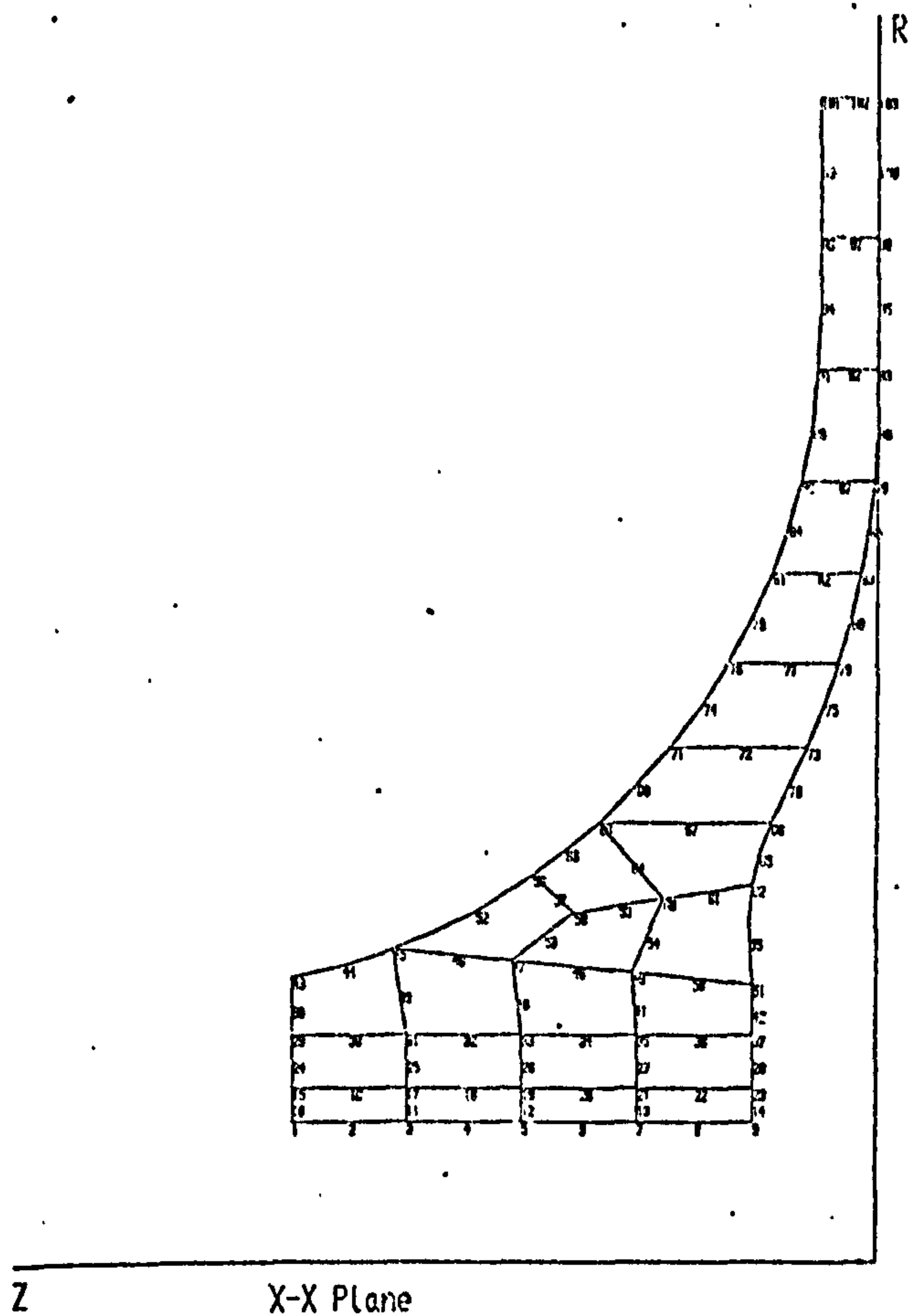


FIG 4-4(b)



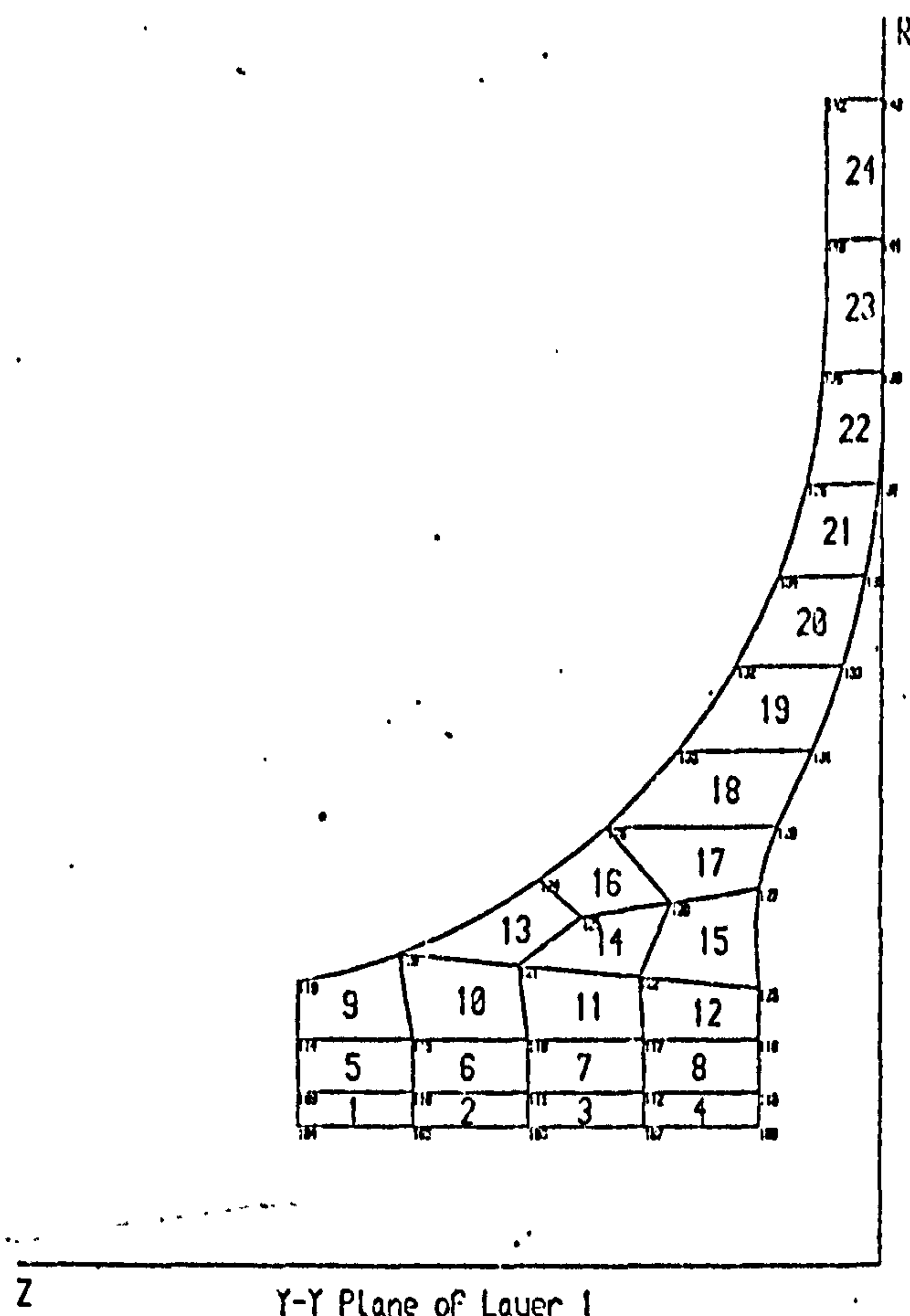
3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS

FIG 4-5(a) IMPELLER MODEL-P MESH III



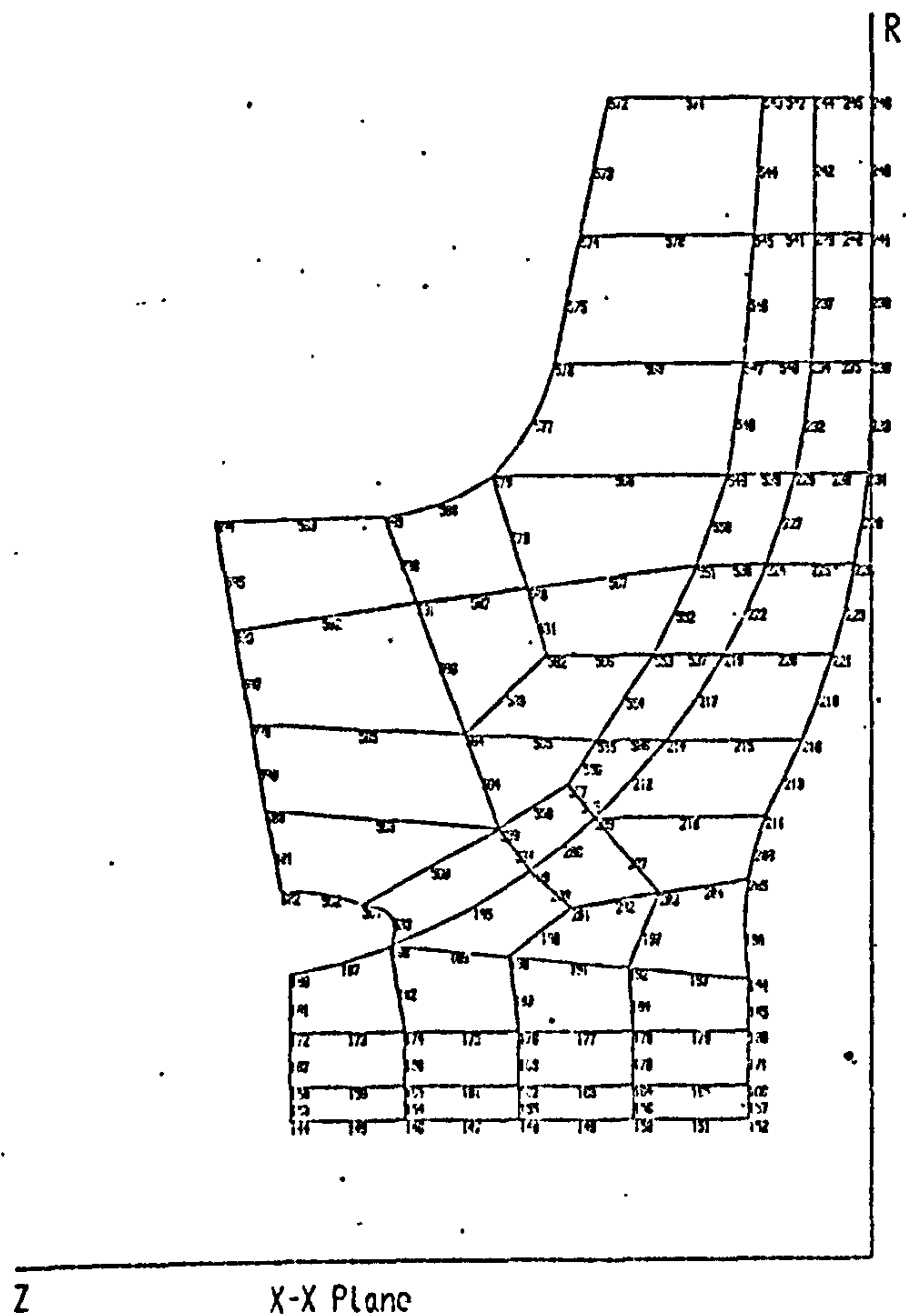
X-X Plane

IMPELLER MODEL-P MESH III



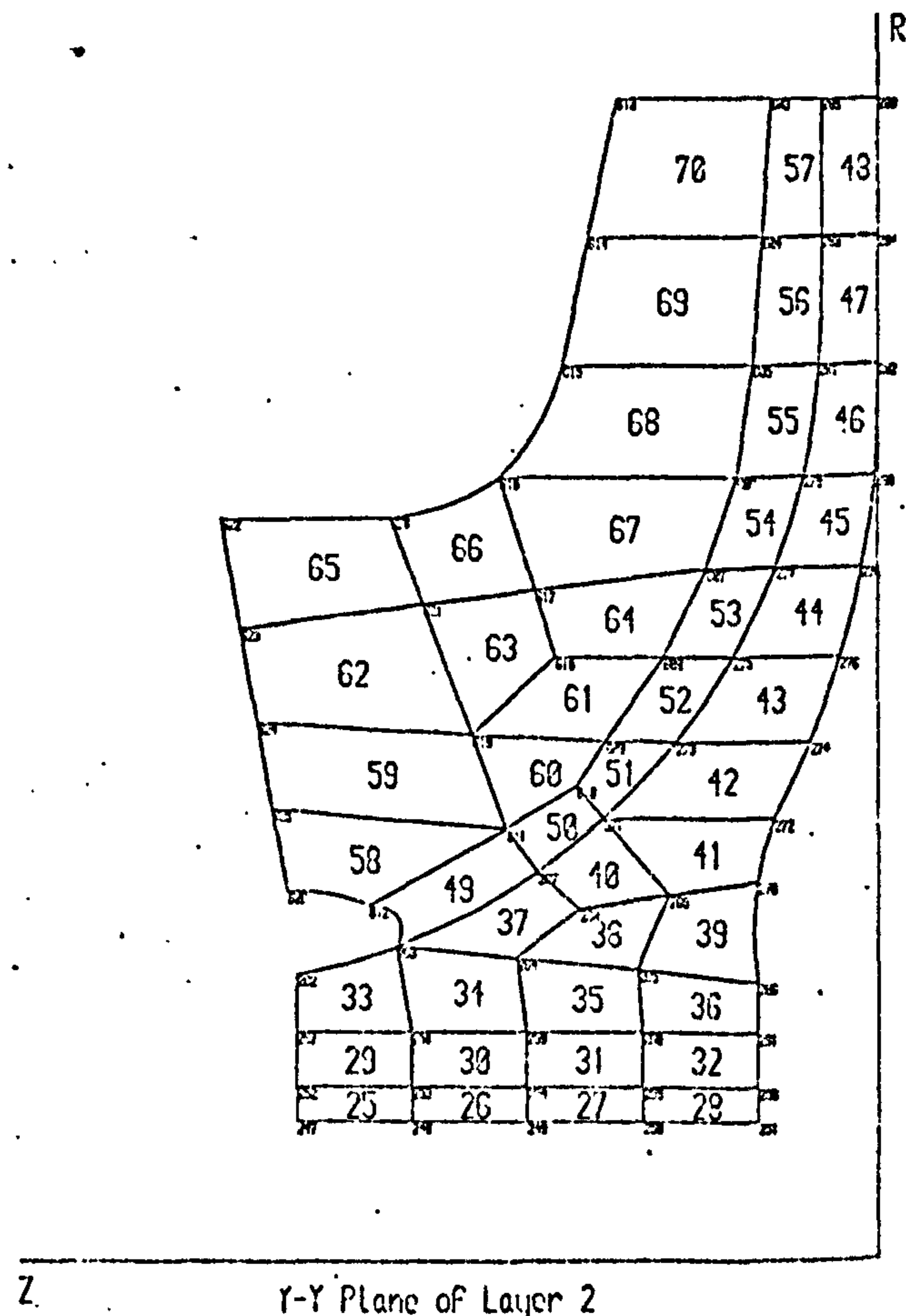
Y-Y Plane of Layer 1

DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS
Sheet 1



X-X Plane

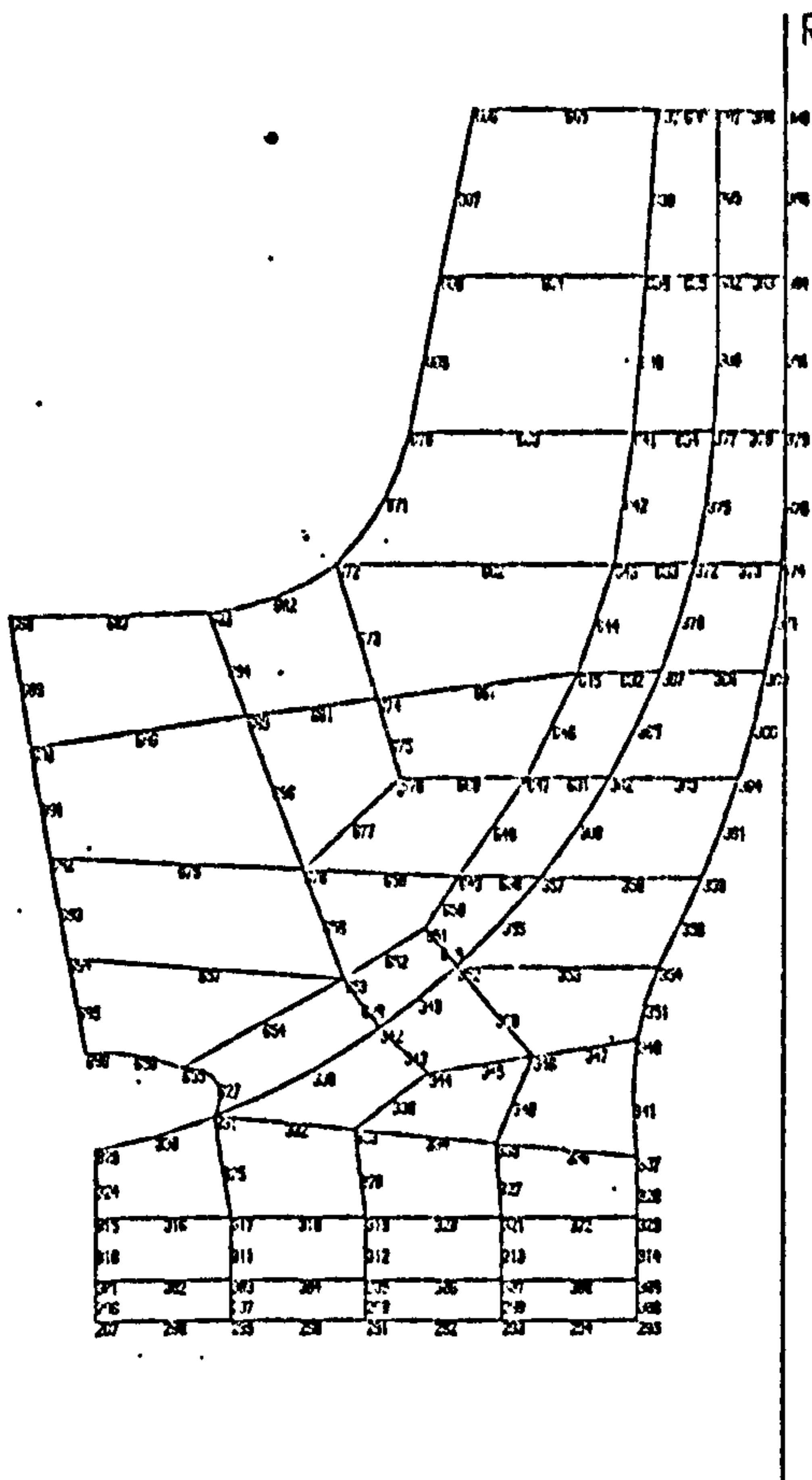
IMPELLER MODEL-P MESH III



Y-Y Plane of Layer 2

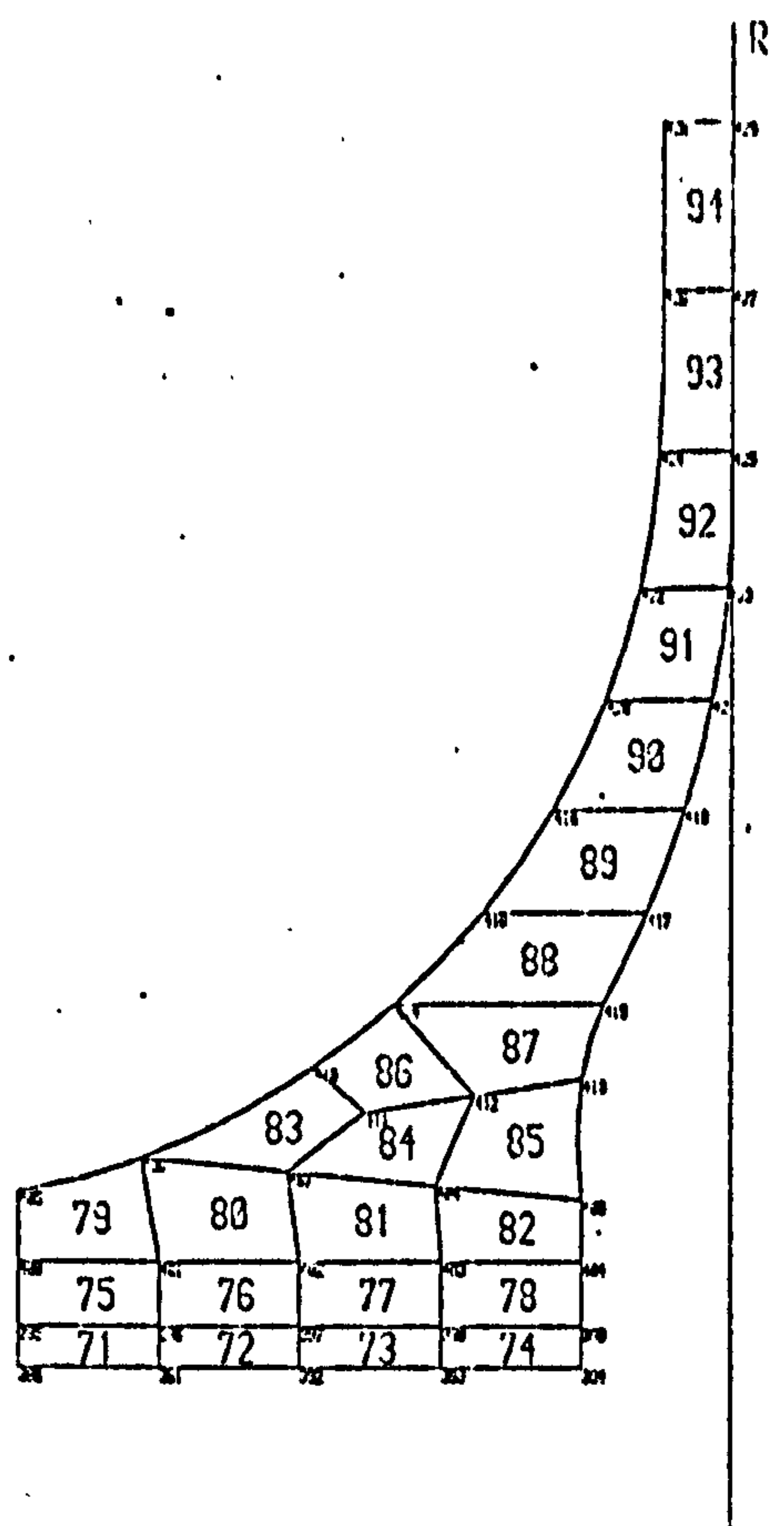
DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS
Sheet 2

**PAGE
NUMBERS
CUT OFF
IN
ORIGINAL**



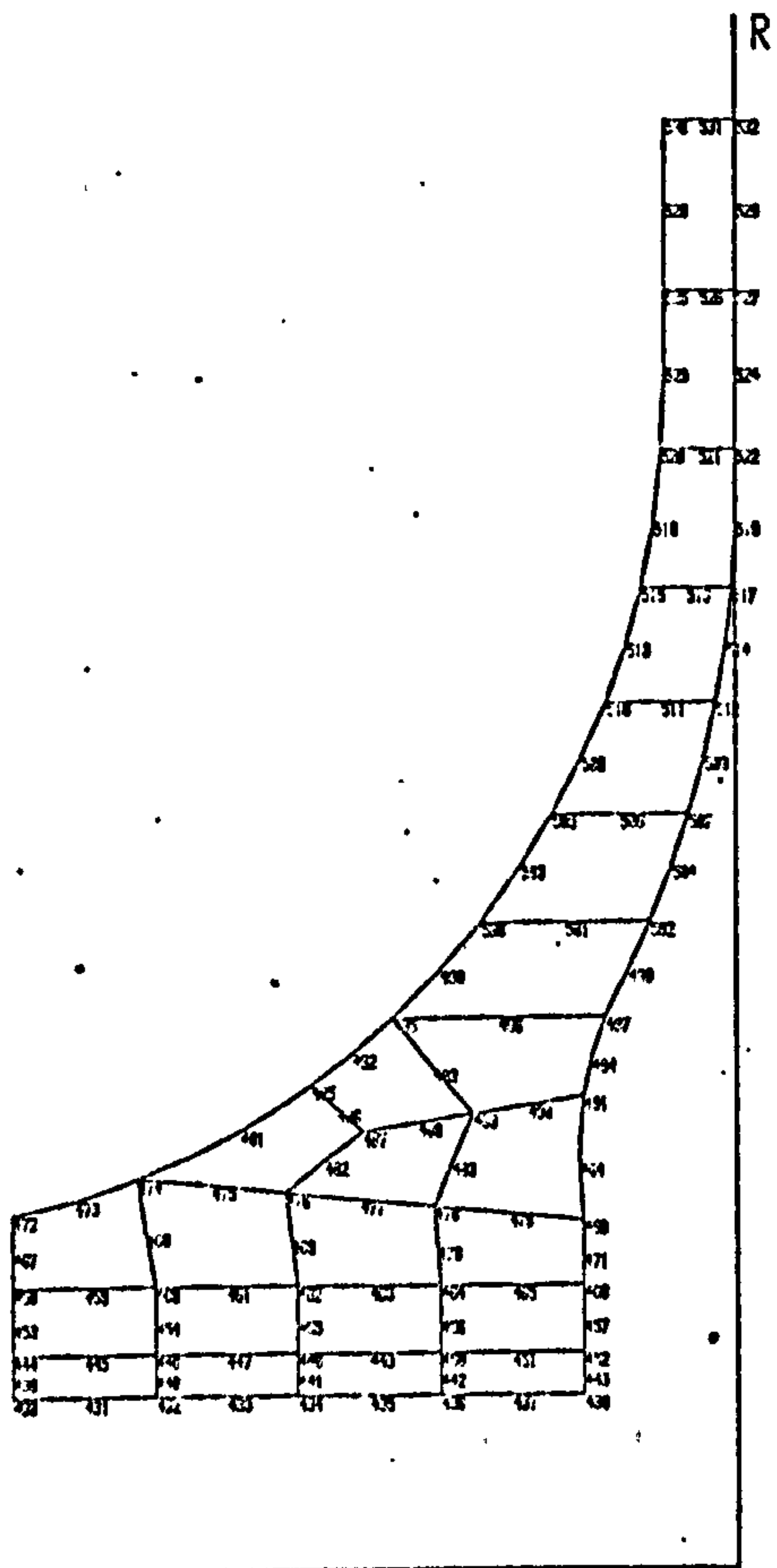
X-X Plane

IMPELLER MODEL-P MESH III



Y-Y Plane of Layer 3

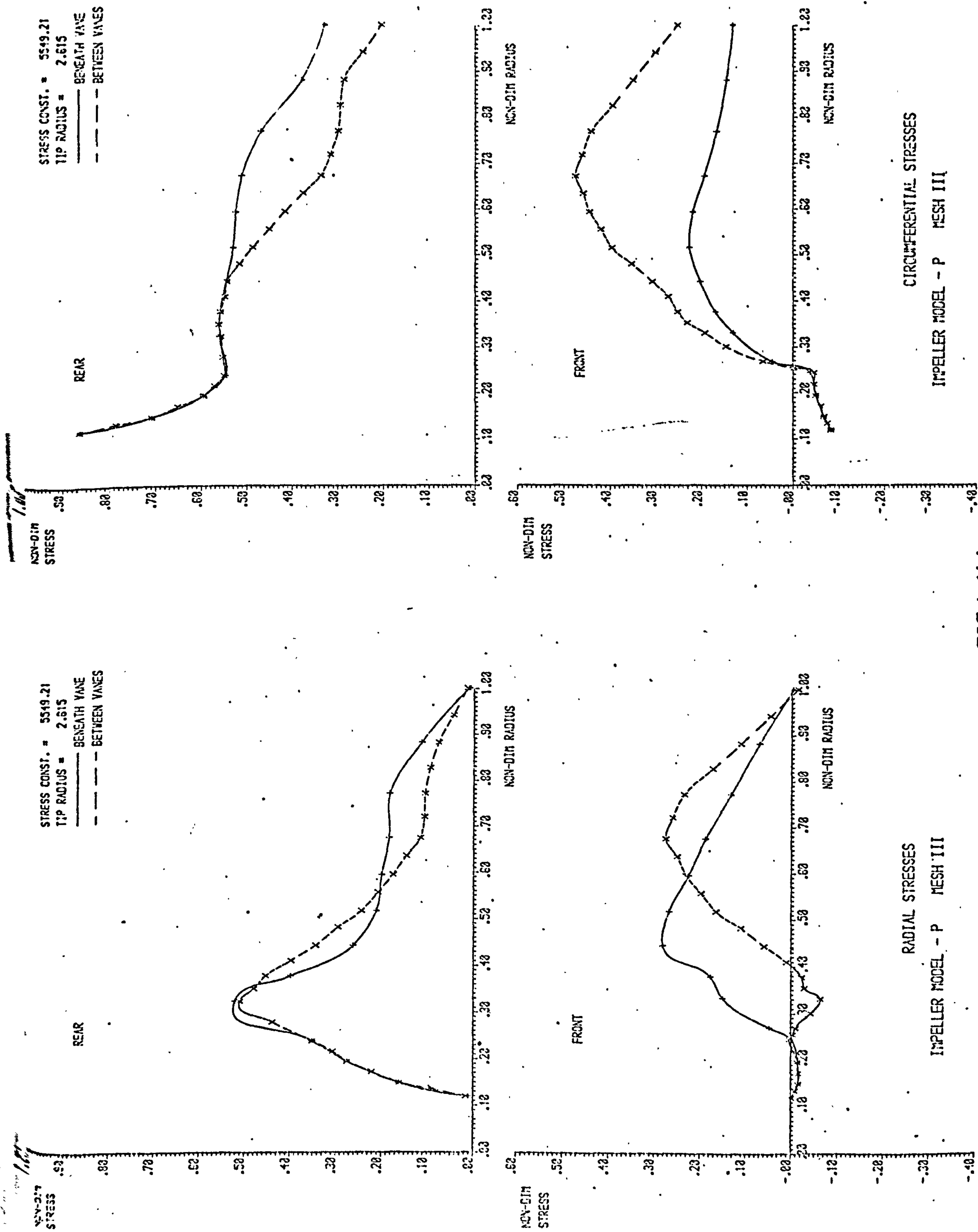
DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS
Sheet 3

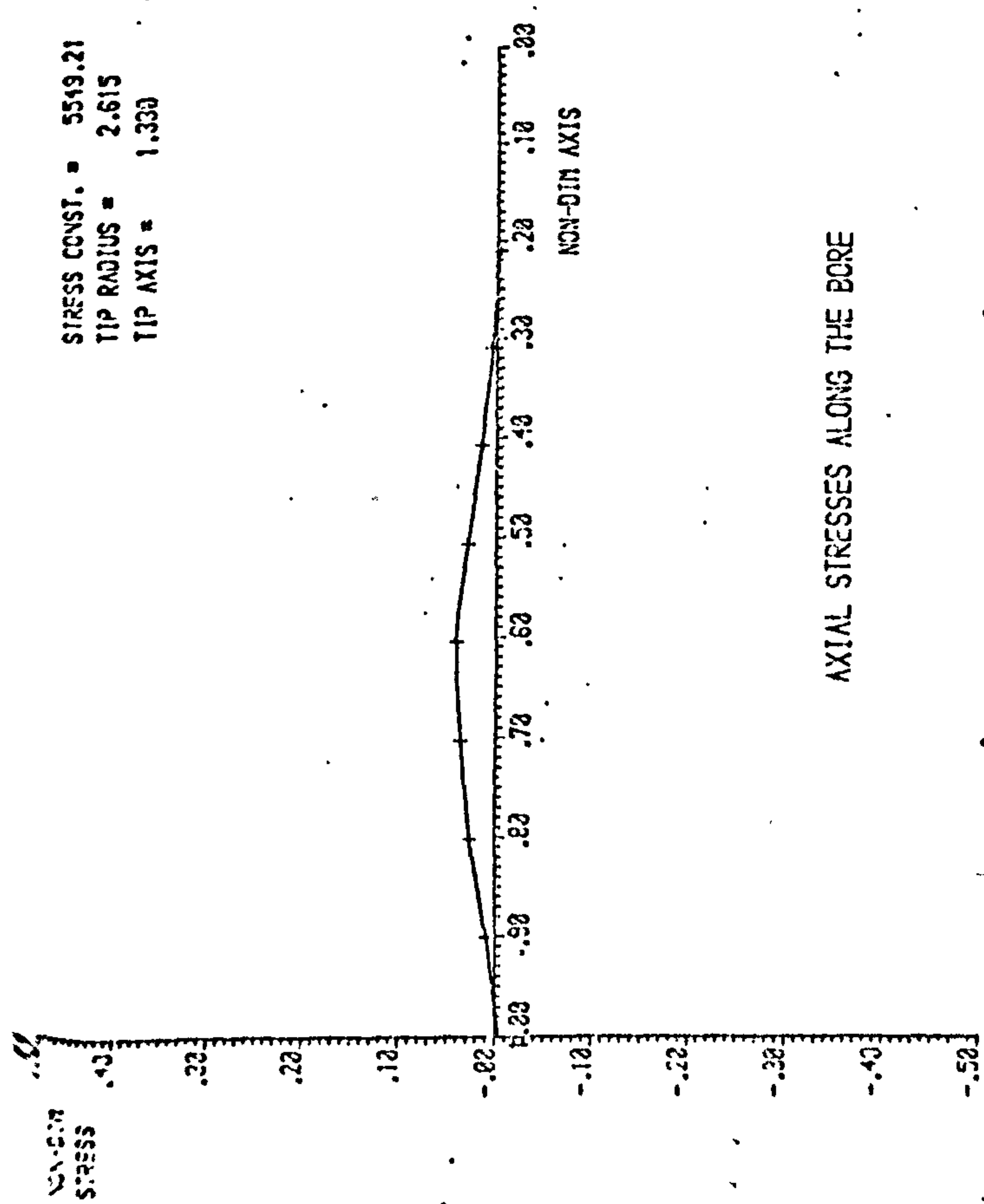
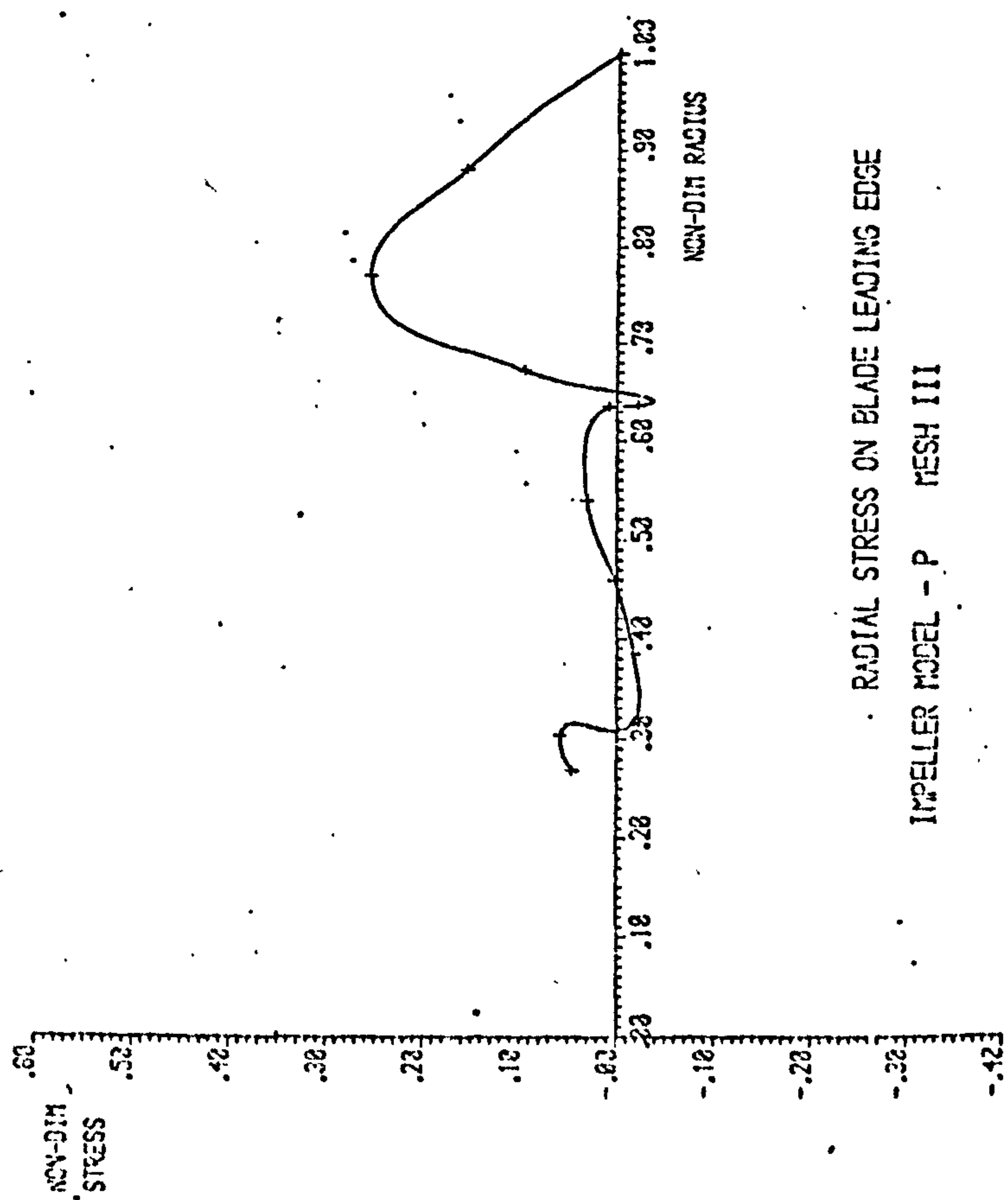
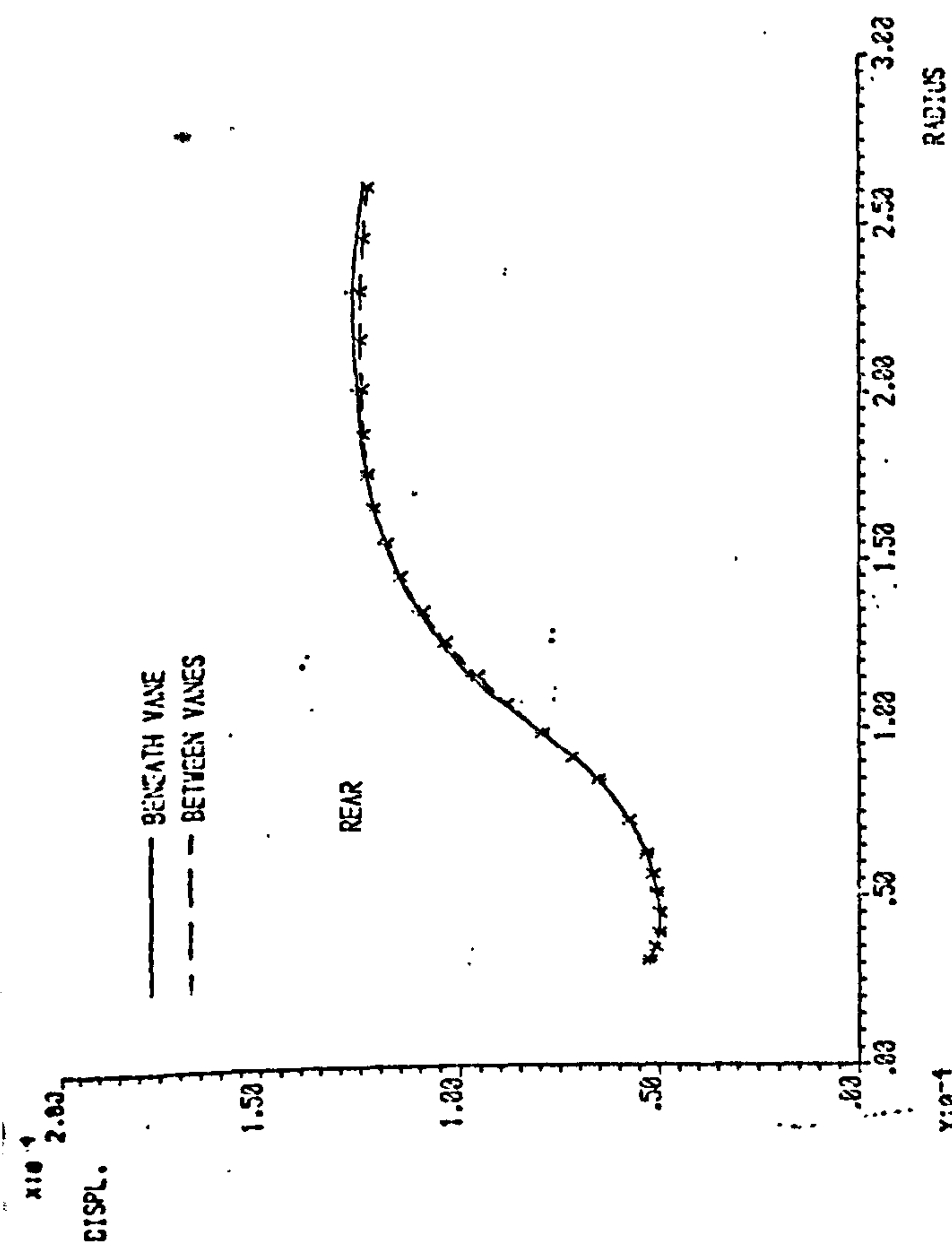
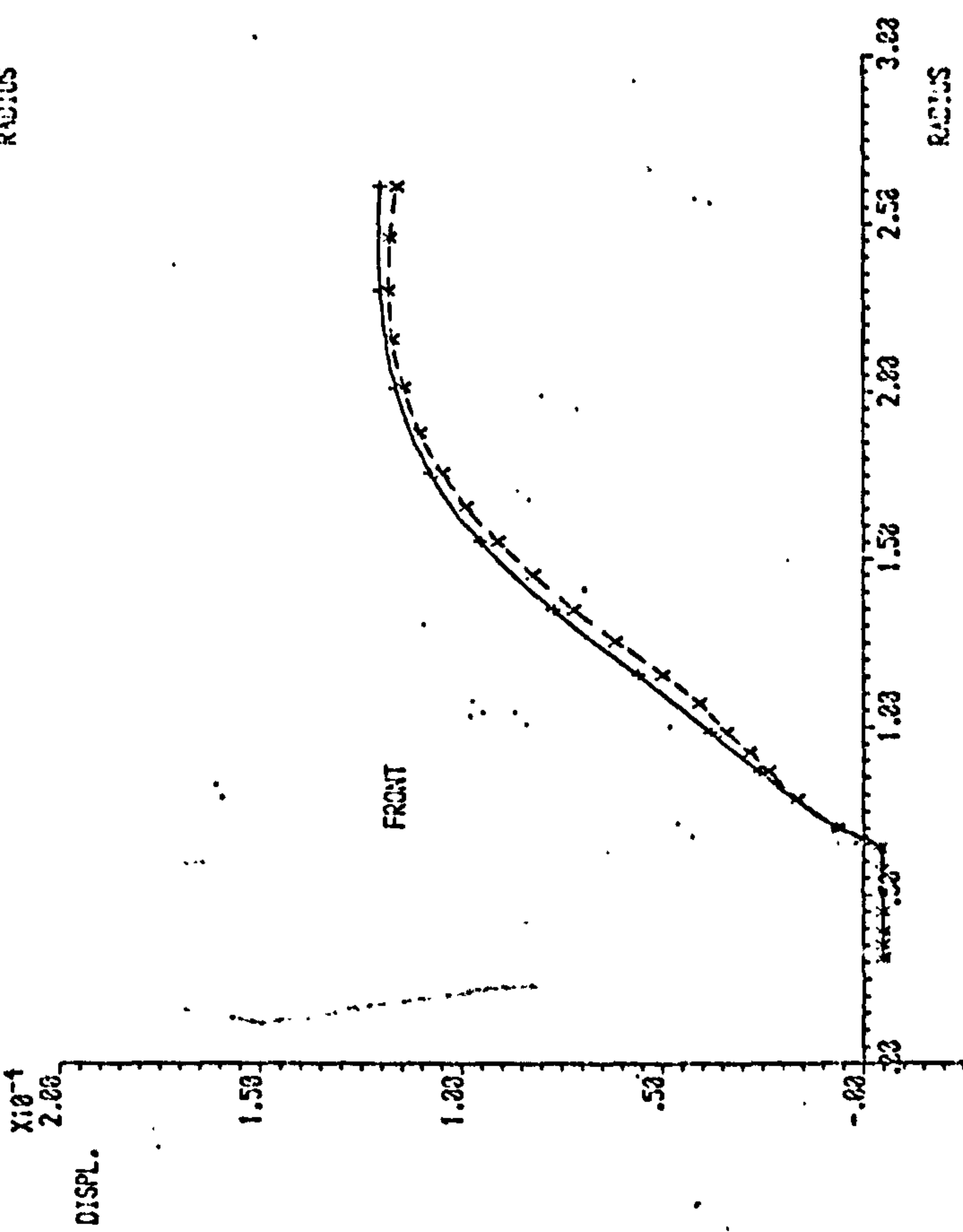


X-X Plane

IMPELLER MODEL-P MESH III

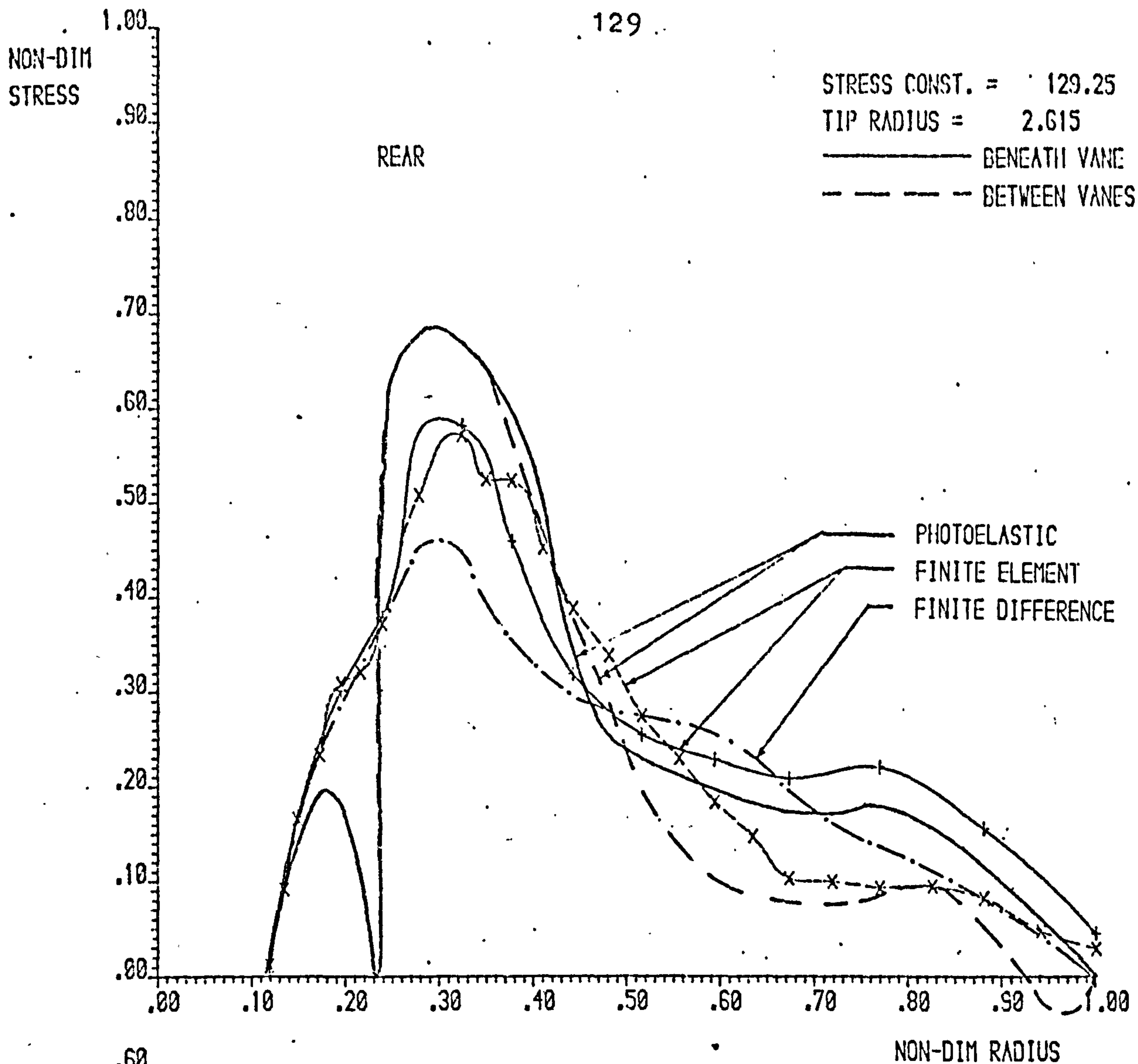
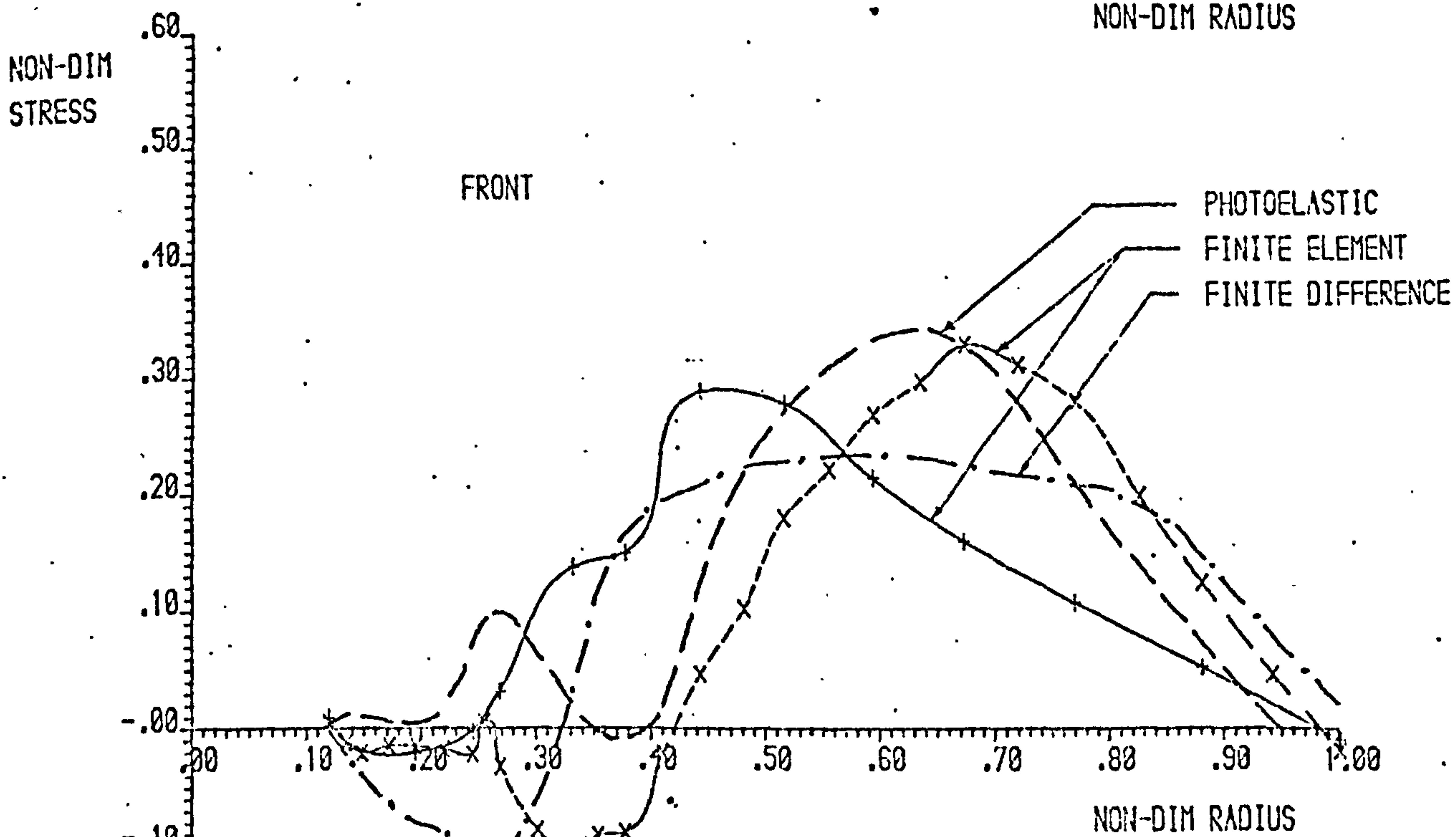
DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS
Sheet 4





RADIAL DISPLACEMENTS
IMPELLER MODEL - P MESH III

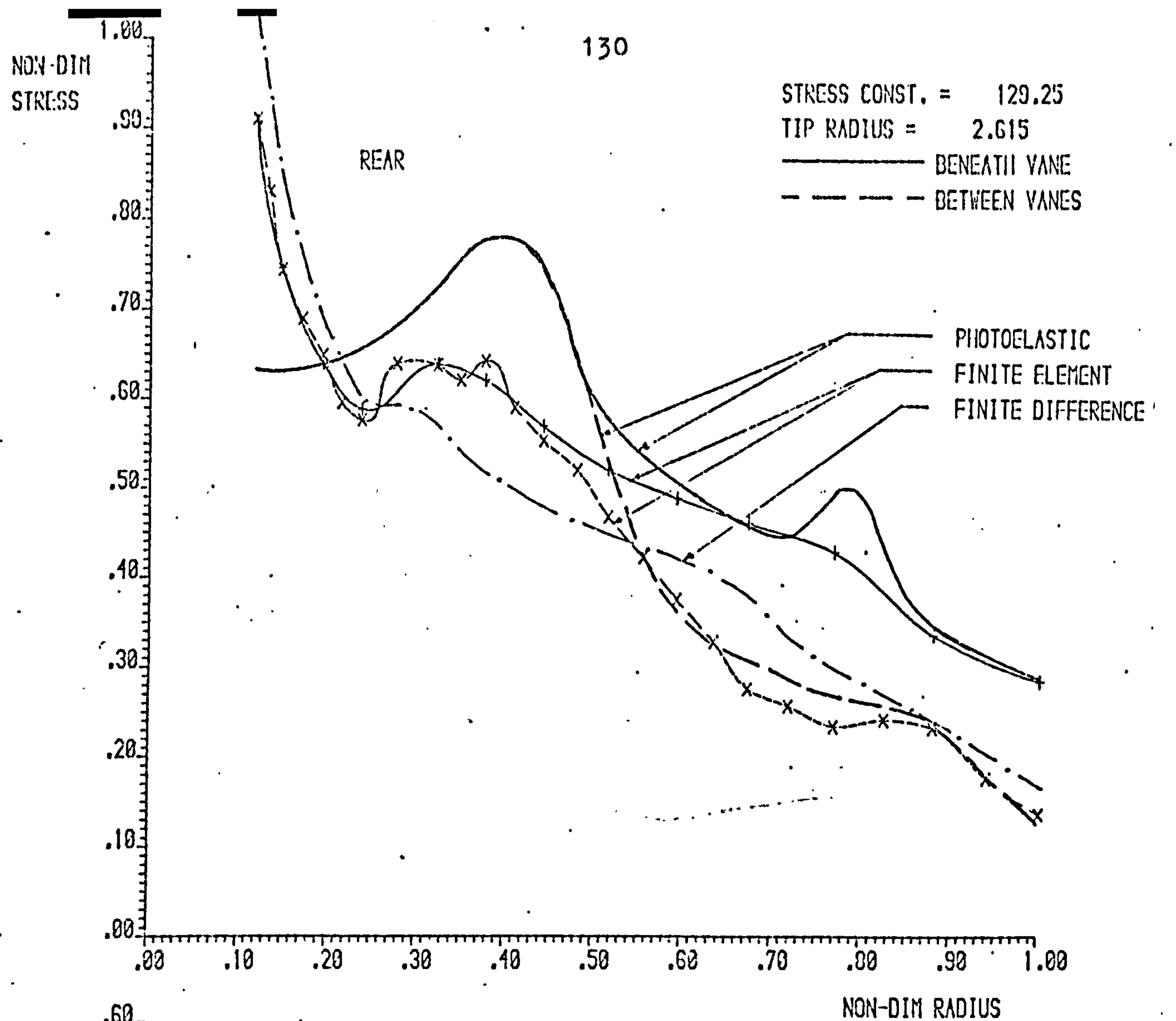
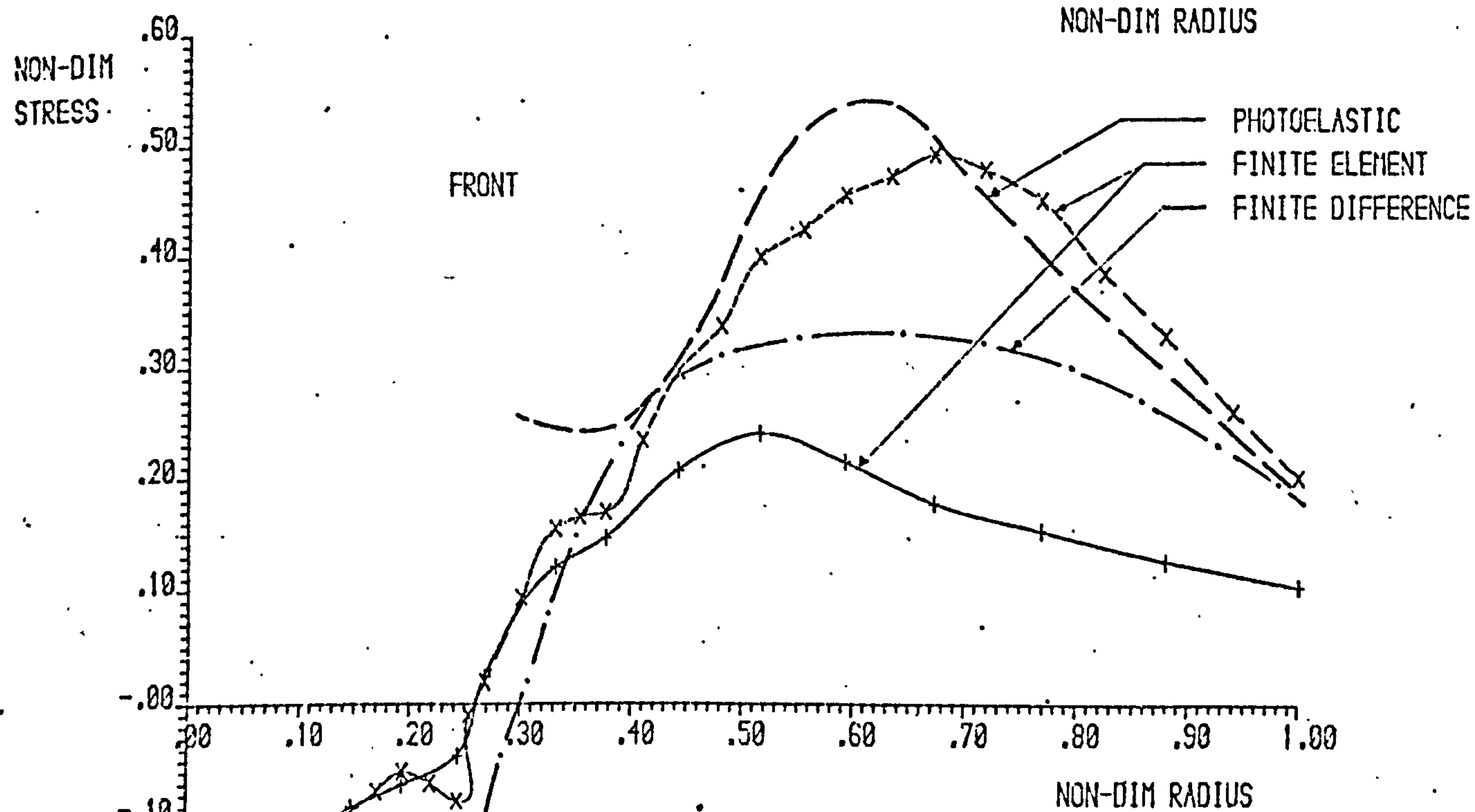
FIG 4-6(b)



RADIAL STRESSES

IMPELLER MODEL - P MESH III

FIG 4-7(a)



CIRCUMFERENTIAL STRESSES

IMPELLER MODEL - P MESH III

FIG 4.7(b)

NON-DIM
STRESS

STRESS CONST. = 129.25
 TIP RADIUS = 2.615
 TIP AXIS = 1.330

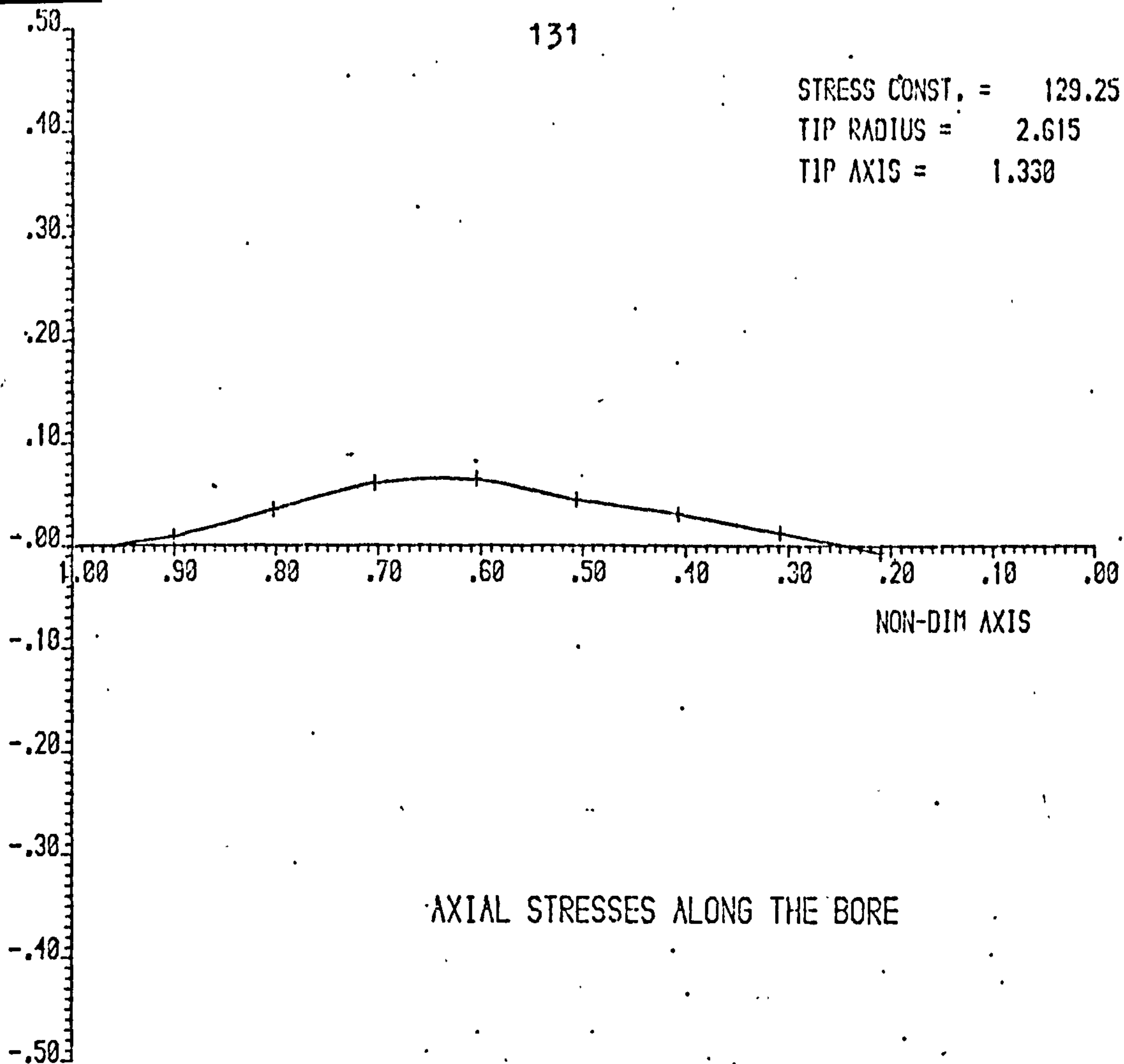
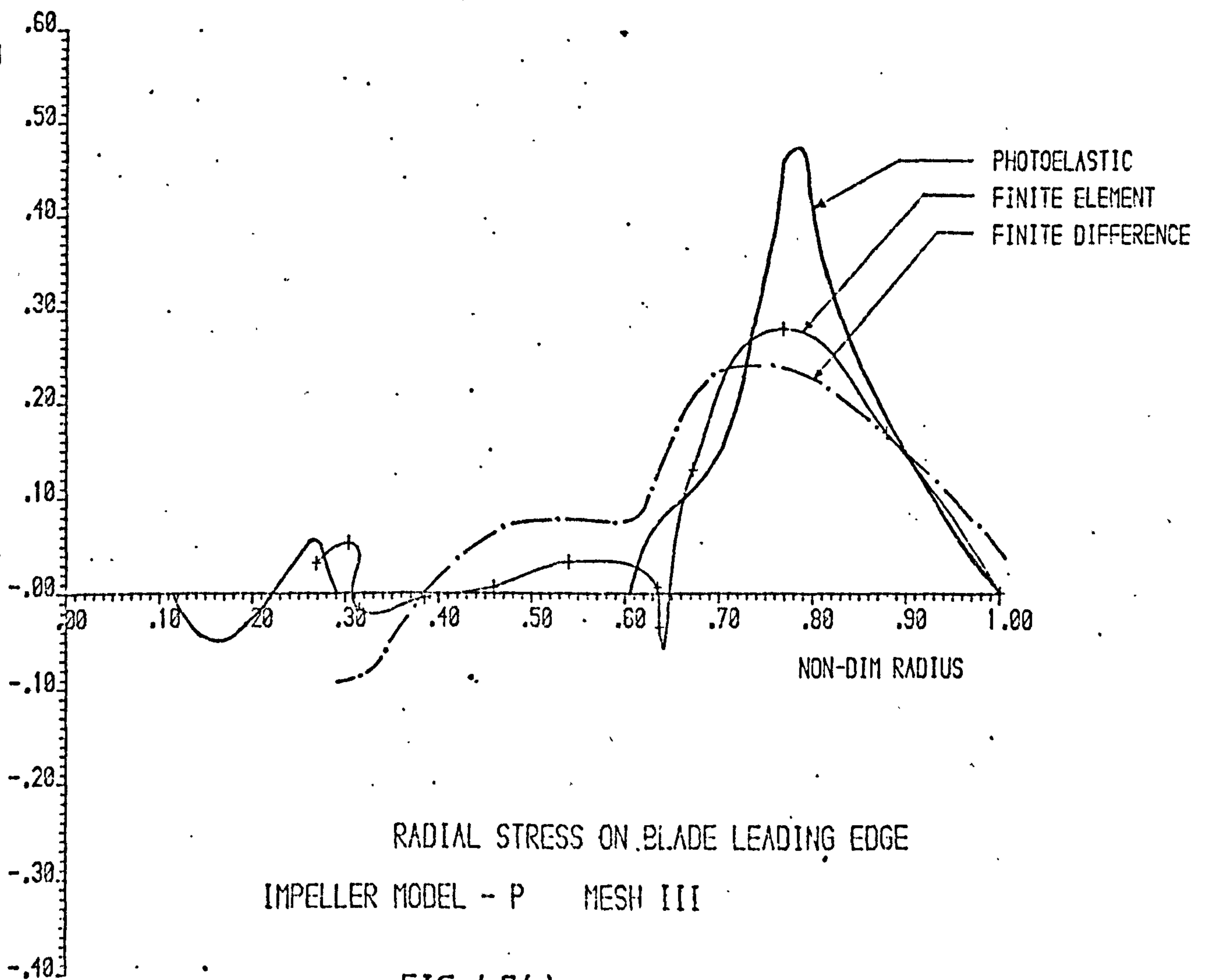
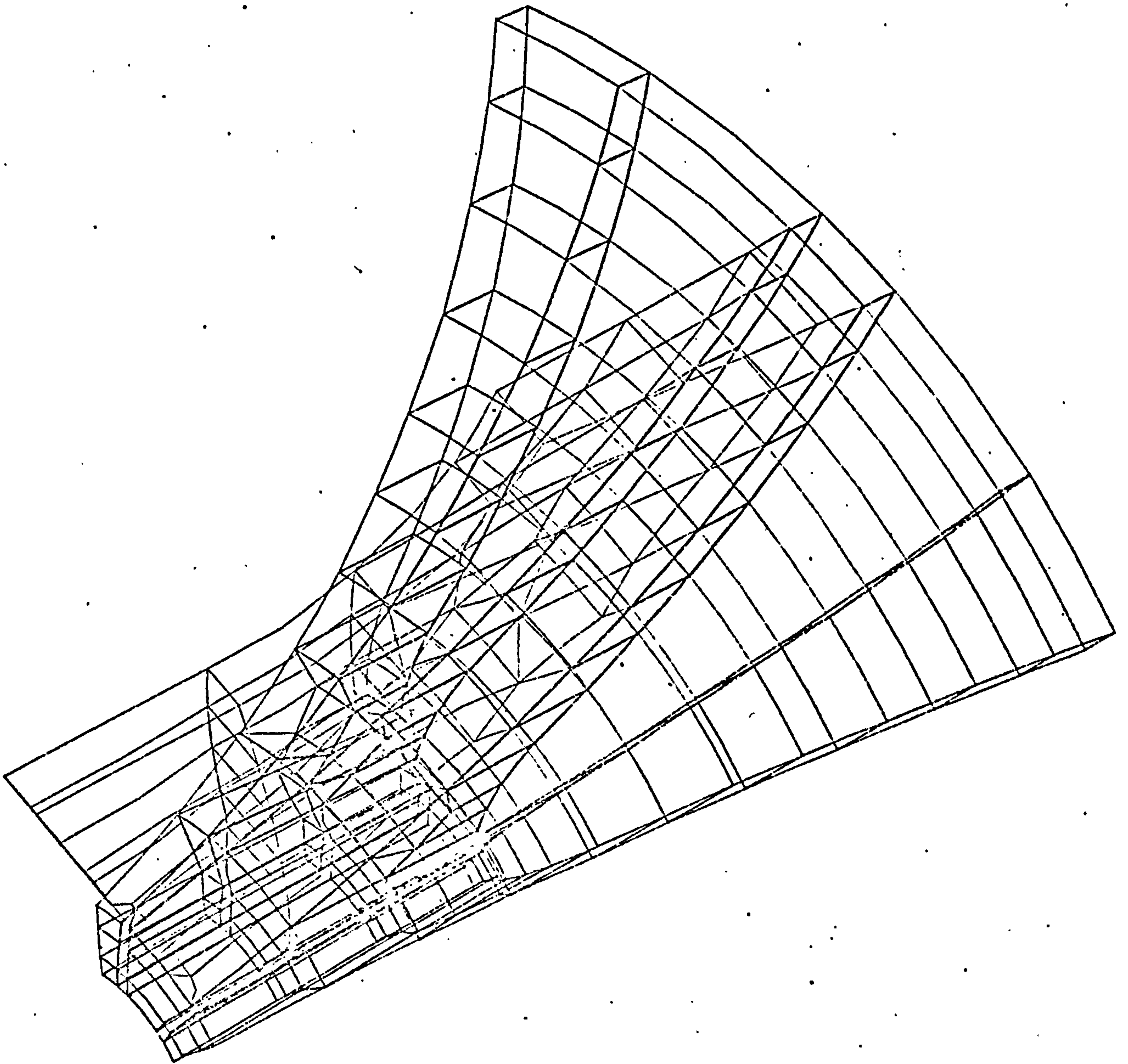
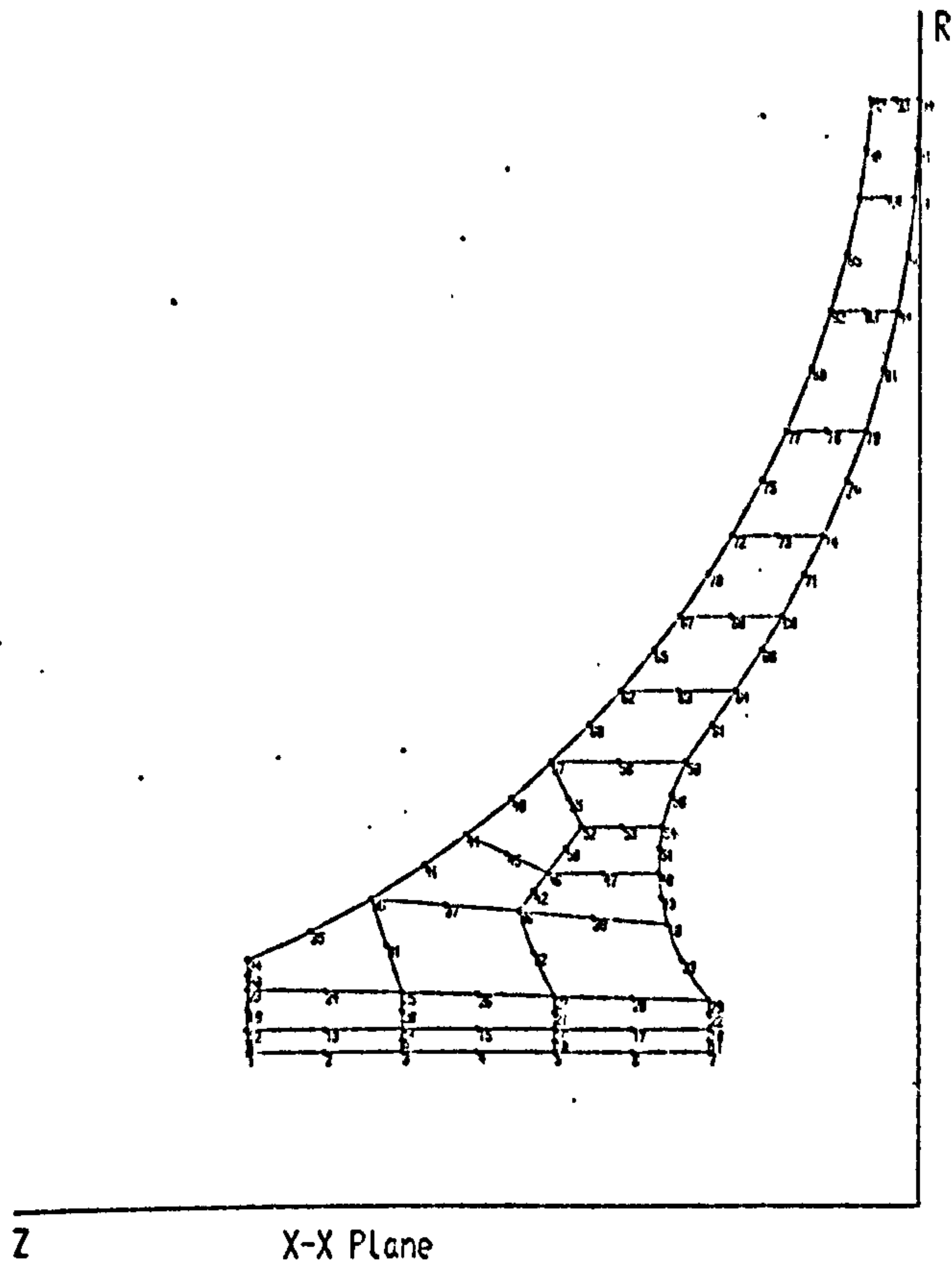
NON-DIM
STRESS

FIG 4-7(c)

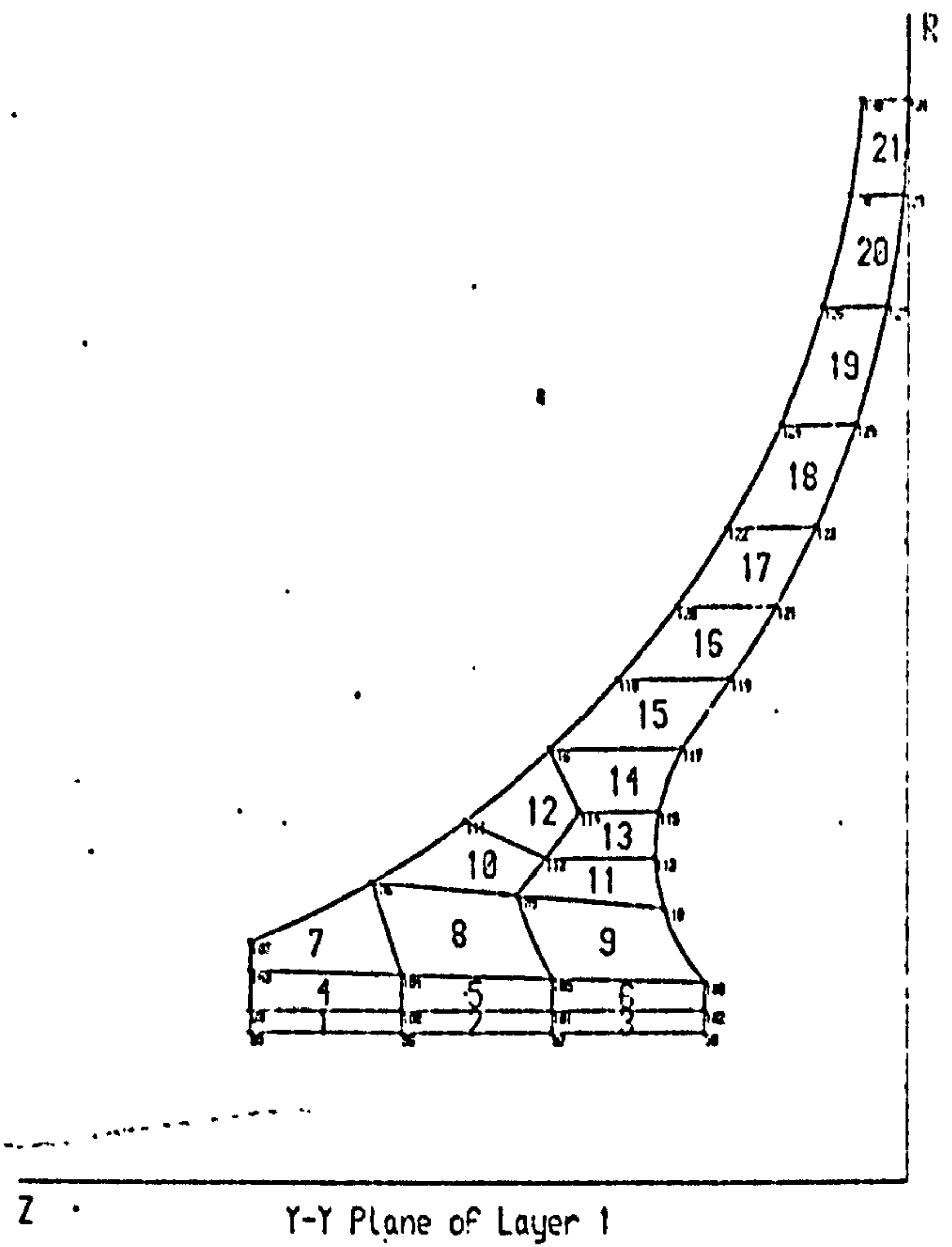


3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN MM SCALE: 40.03 TIMES THE SUPPLIED UNITS

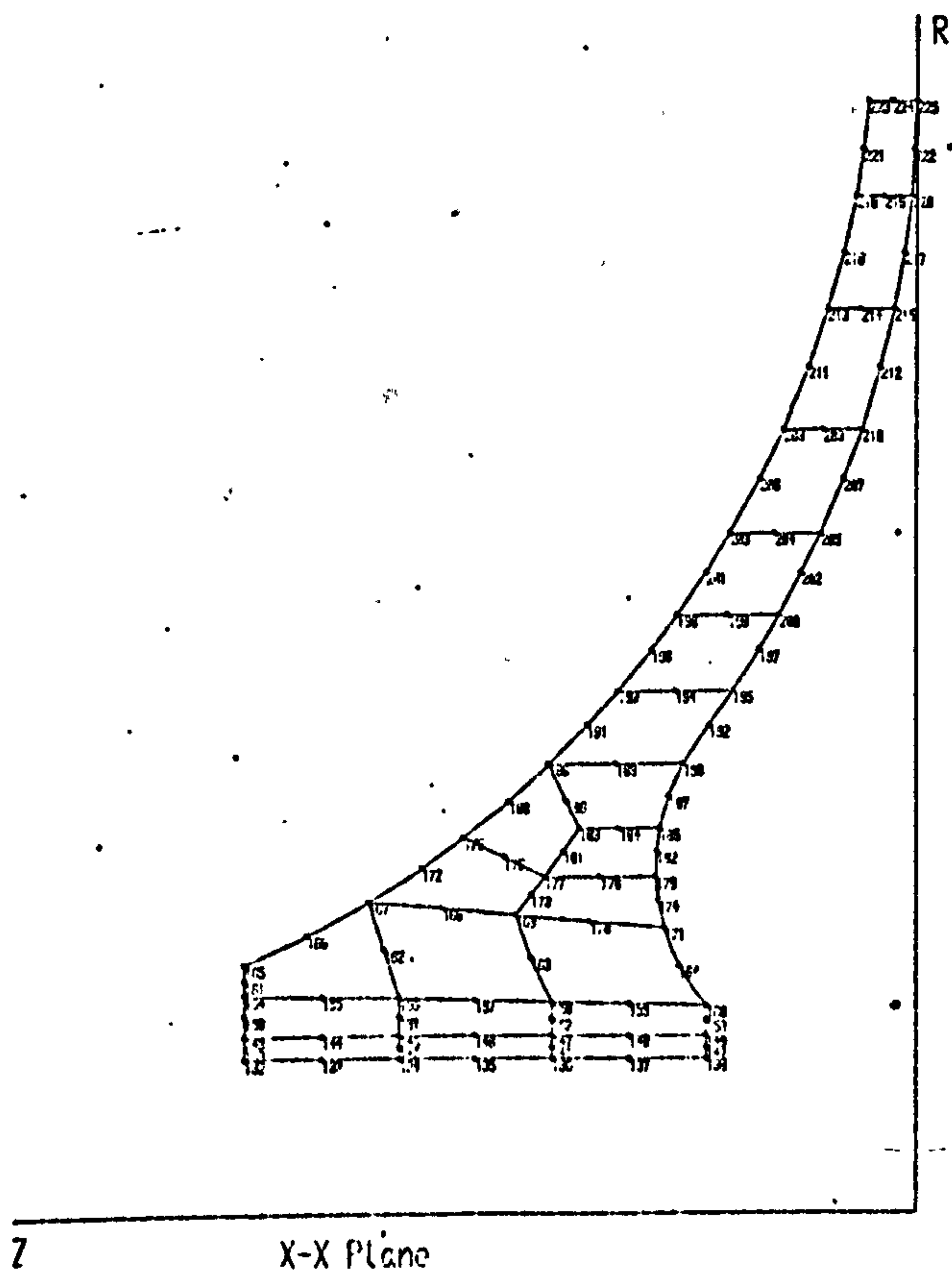
FIG 4-8(a) IMPELLER MODEL-N NESH III



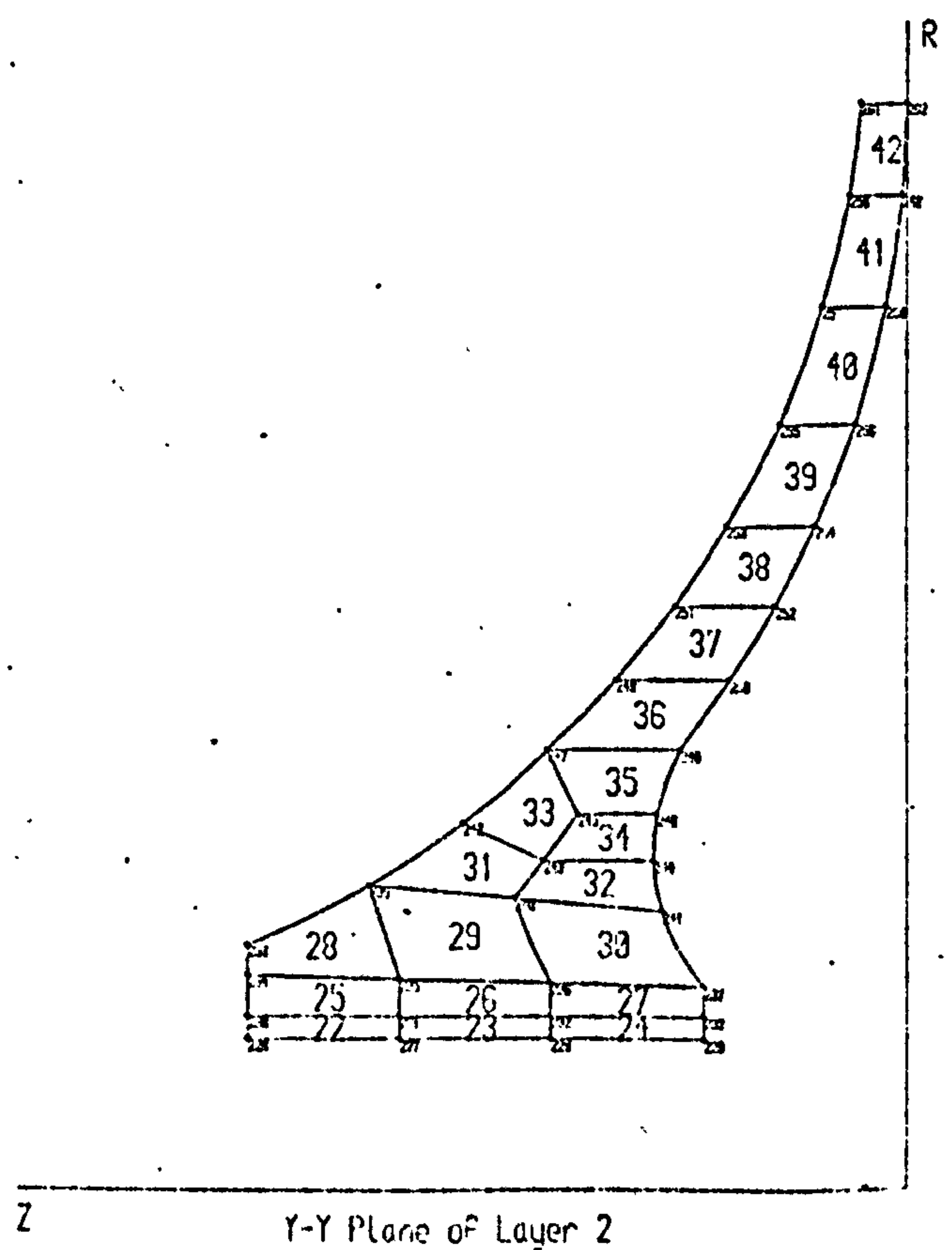
IMPELLER MODEL-N MESH III



DRAWN IN MM SCALE: 40.00 TIMES THE SUPPLIED UNITS
Sheet 1

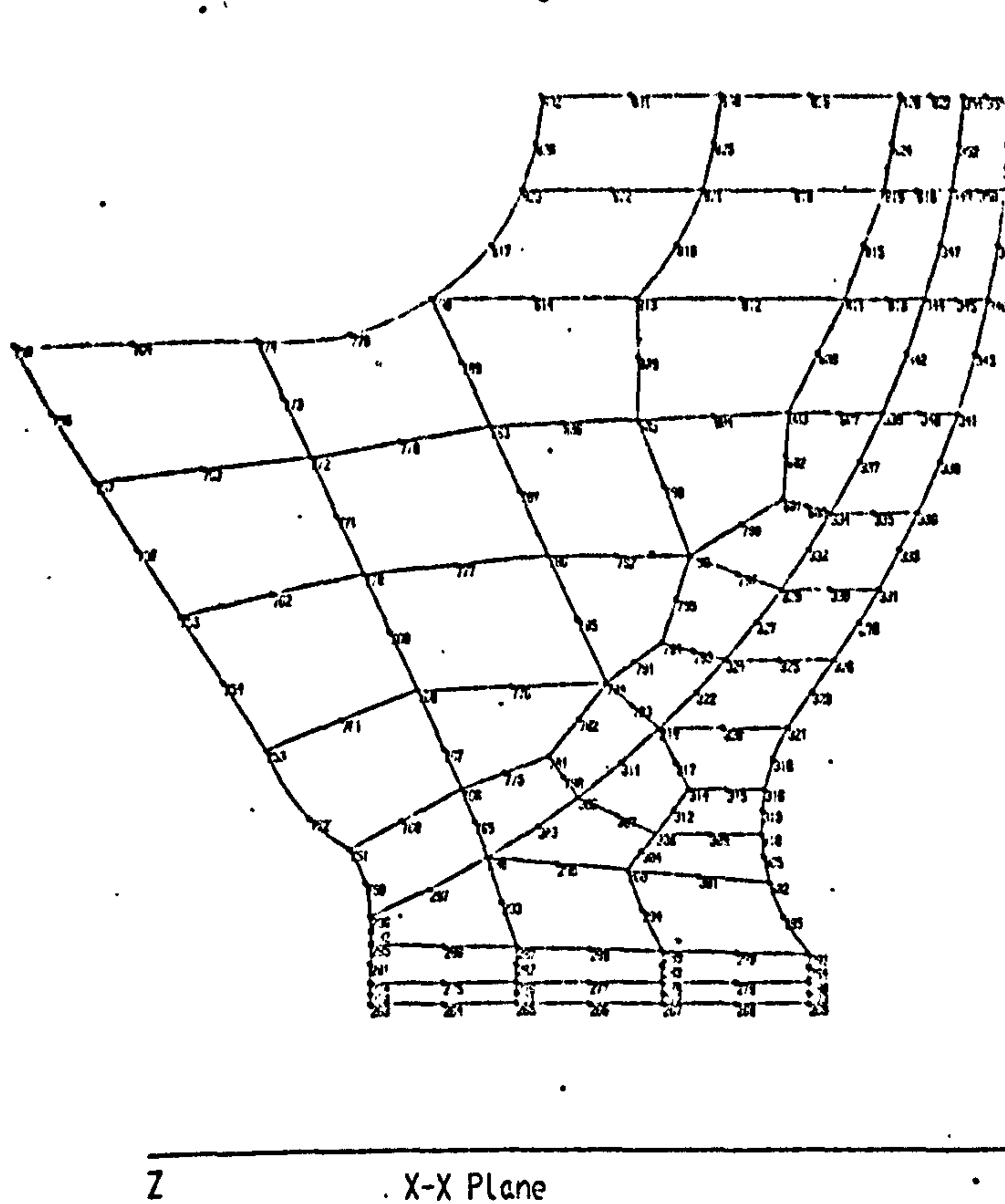


IMPELLER MODEL-N MESH III

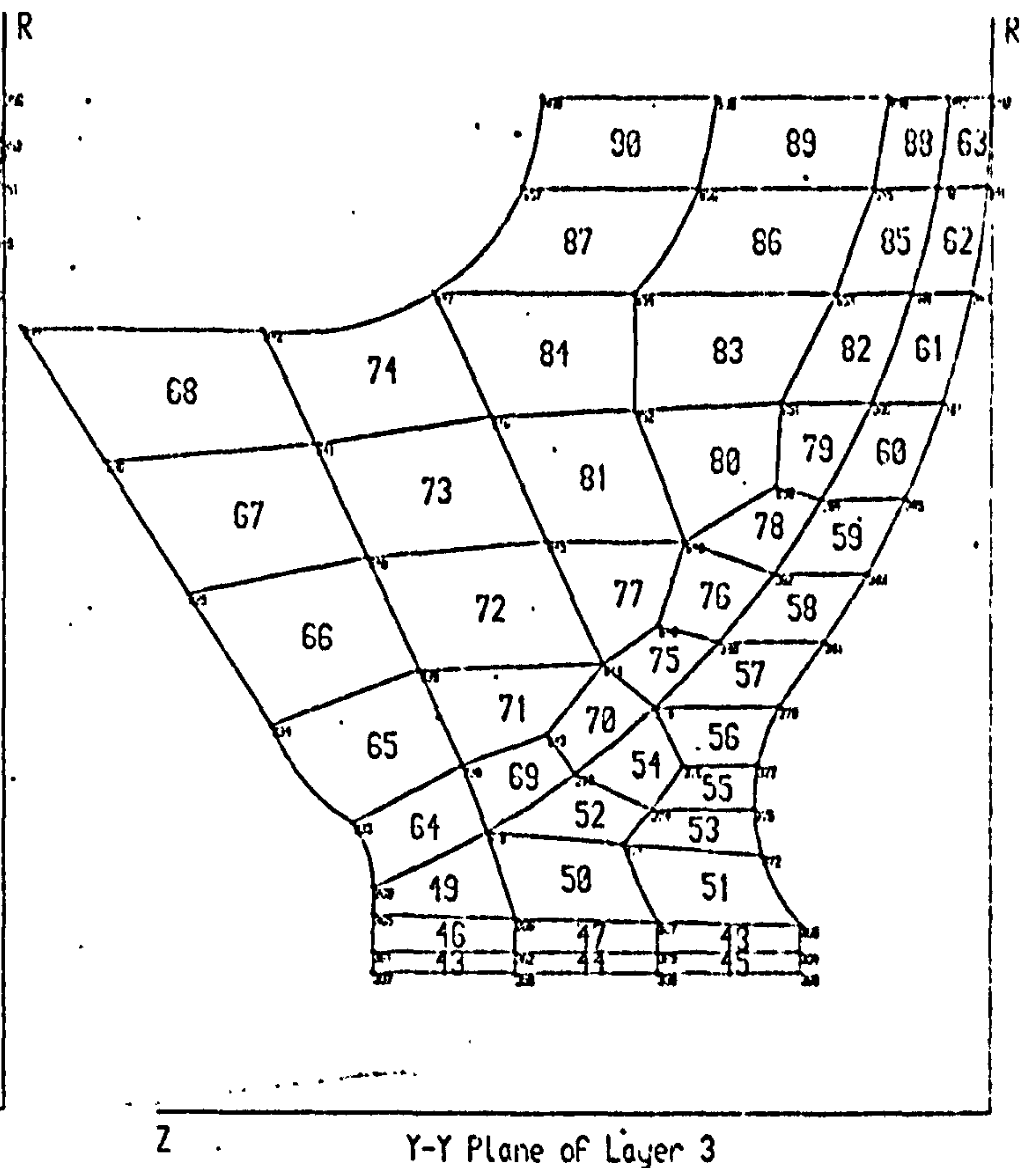


DRAWN IN MM SCALE: 40.00 TIMES THE SUPPLIED UNITS
Sheet 2

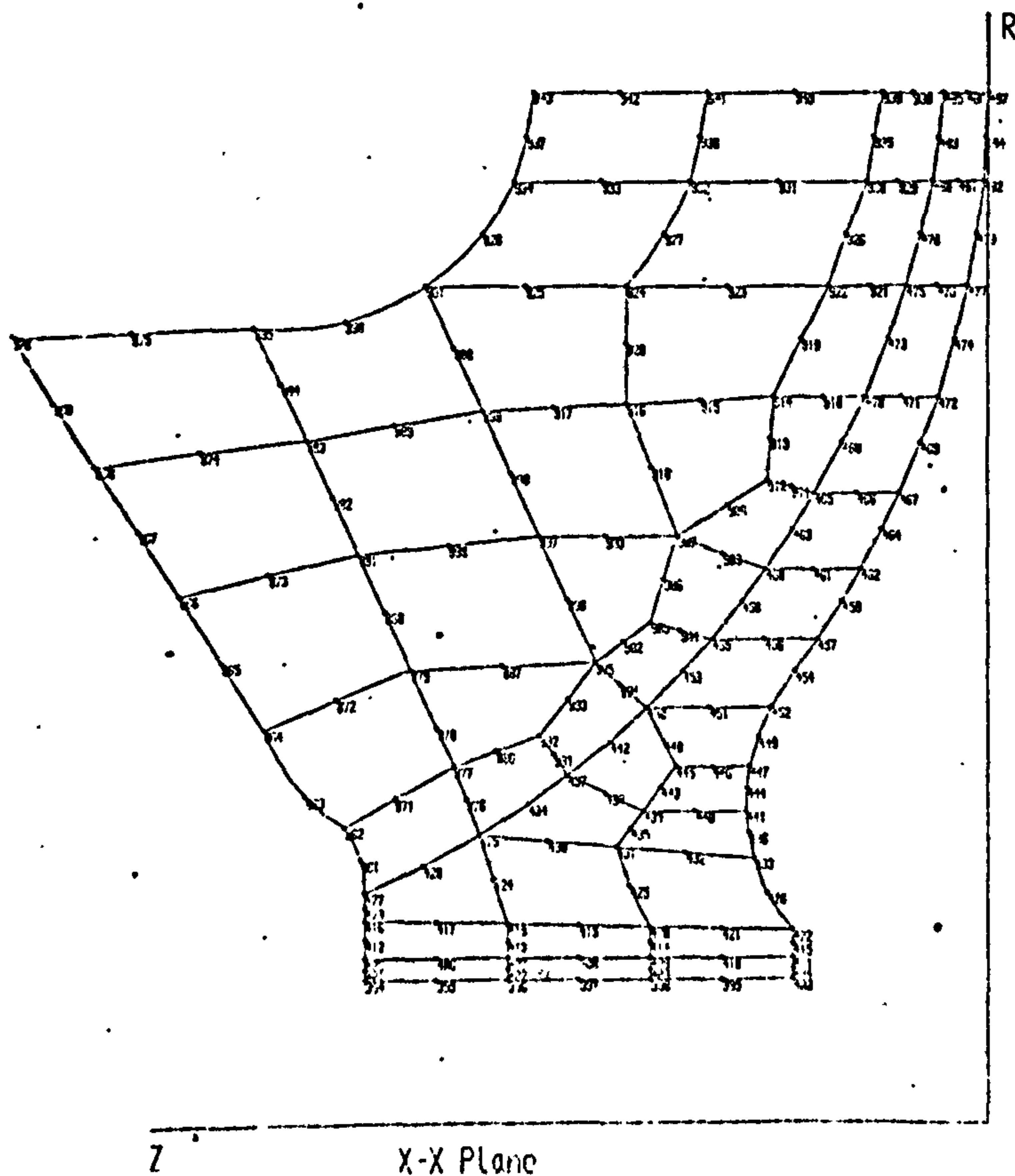
FIG 4-8(b)



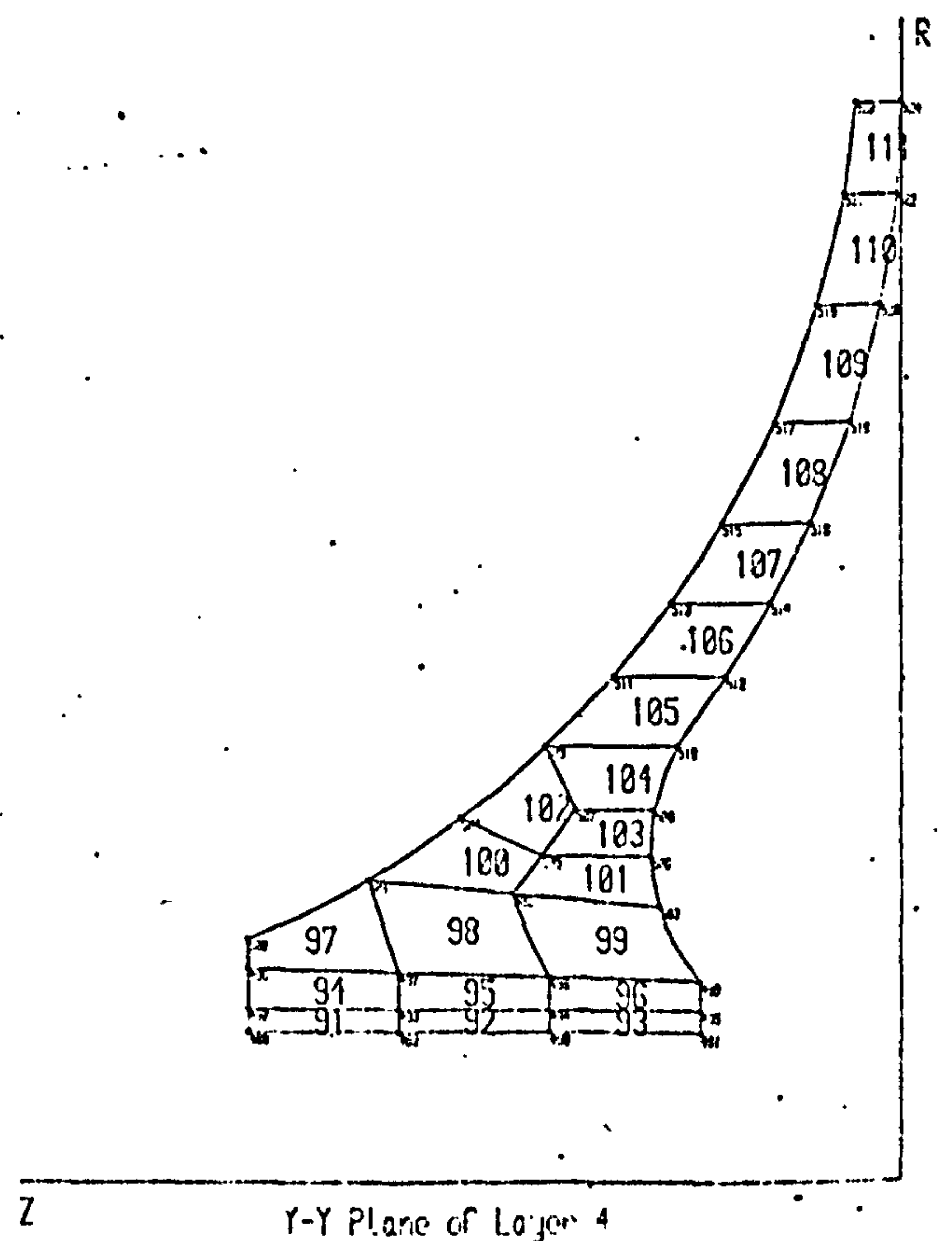
IMPELLER MODEL-N MESH III



DRAWN IN MM SCALE: 40.00 TIMES THE SUPPLIED UNITS
Sheet 3

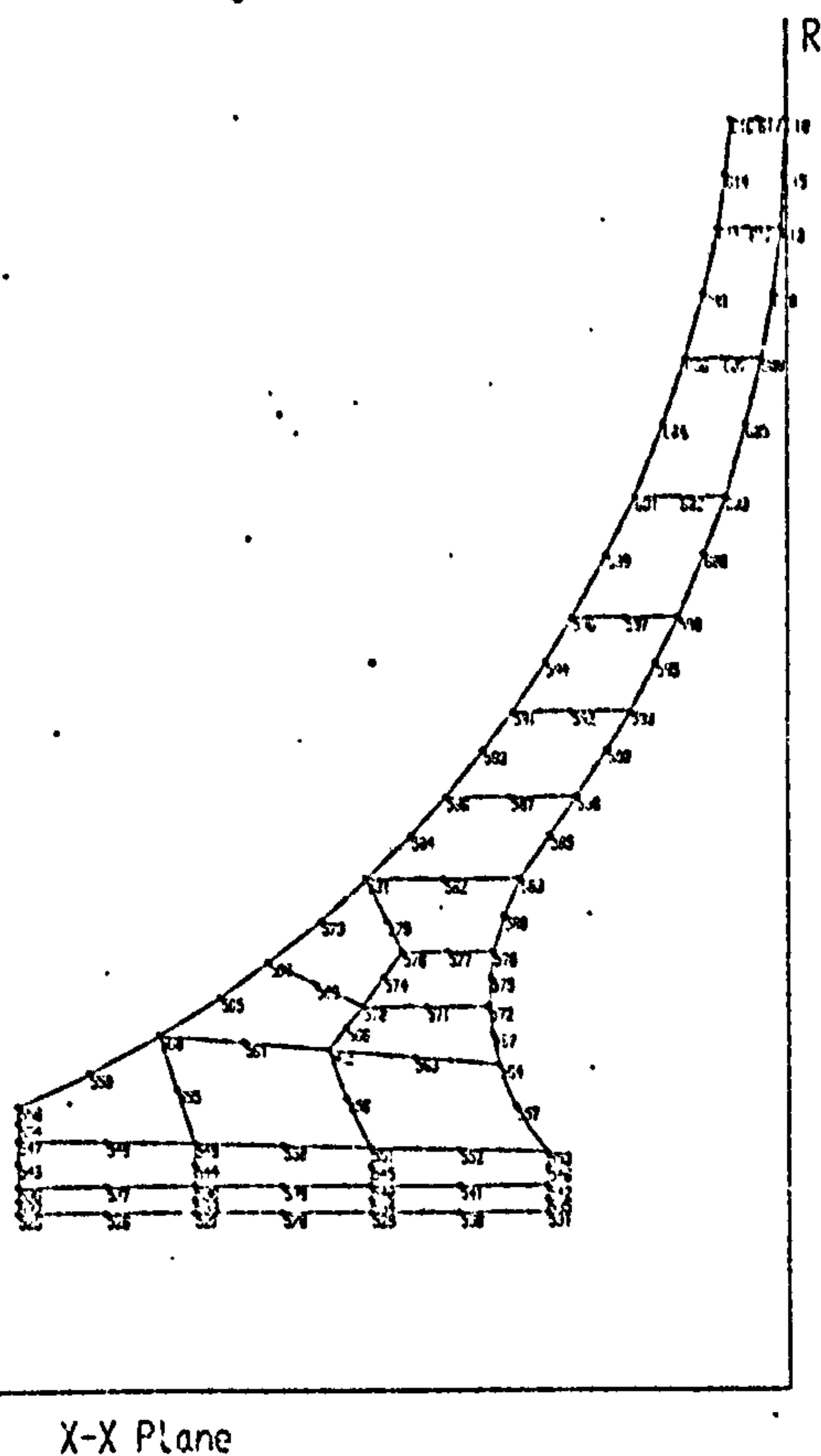


IMPELLER MODEL-N MESH III



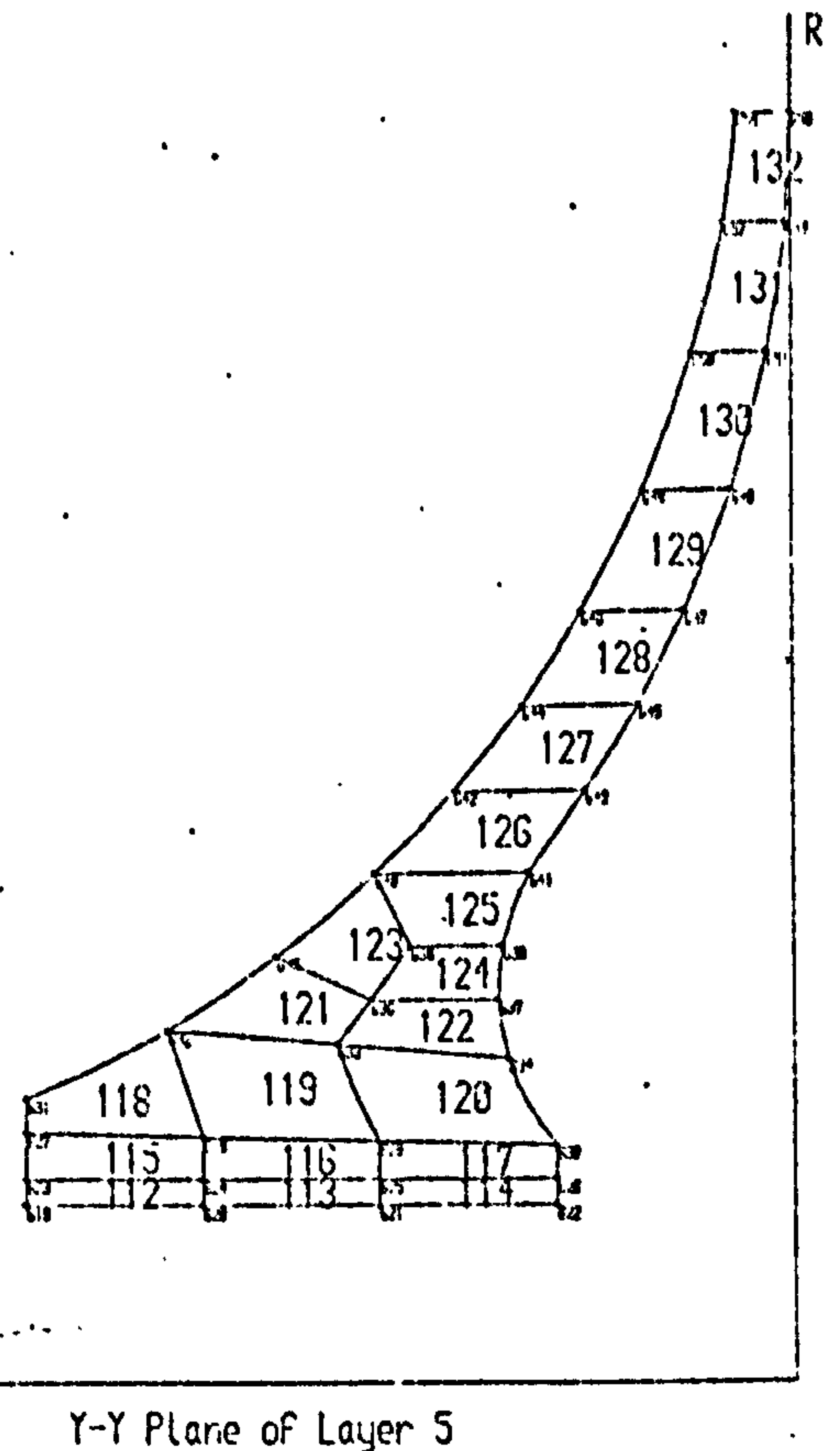
DRAWN IN MM SCALE: 40.00 TIMES THE SUPPLIED UNITS
Sheet 4

FIG 4-8(c)

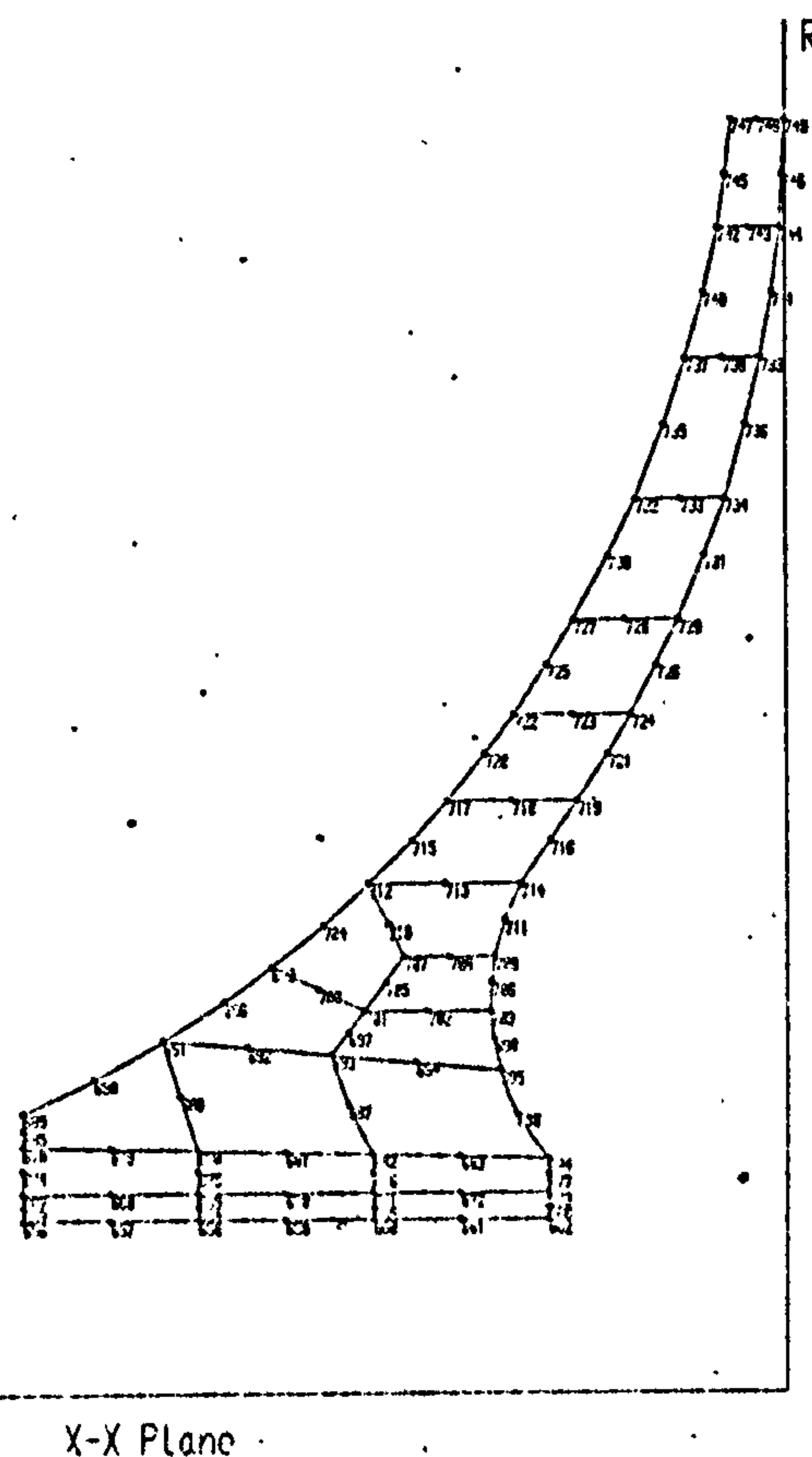


X-X Plane

IMPELLER MODEL-N MESH III



Y-Y Plane of Layer 5

DRAWN IN MM SCALE: 40.00 TIMES THE SUPPLIED UNITS
Sheet 5

X-X Plane

IMPELLER MODEL-N MESH III

DRAWN IN MM SCALE: 40.00 TIMES THE SUPPLIED UNITS
Sheet 6

FIG 4-8(d)

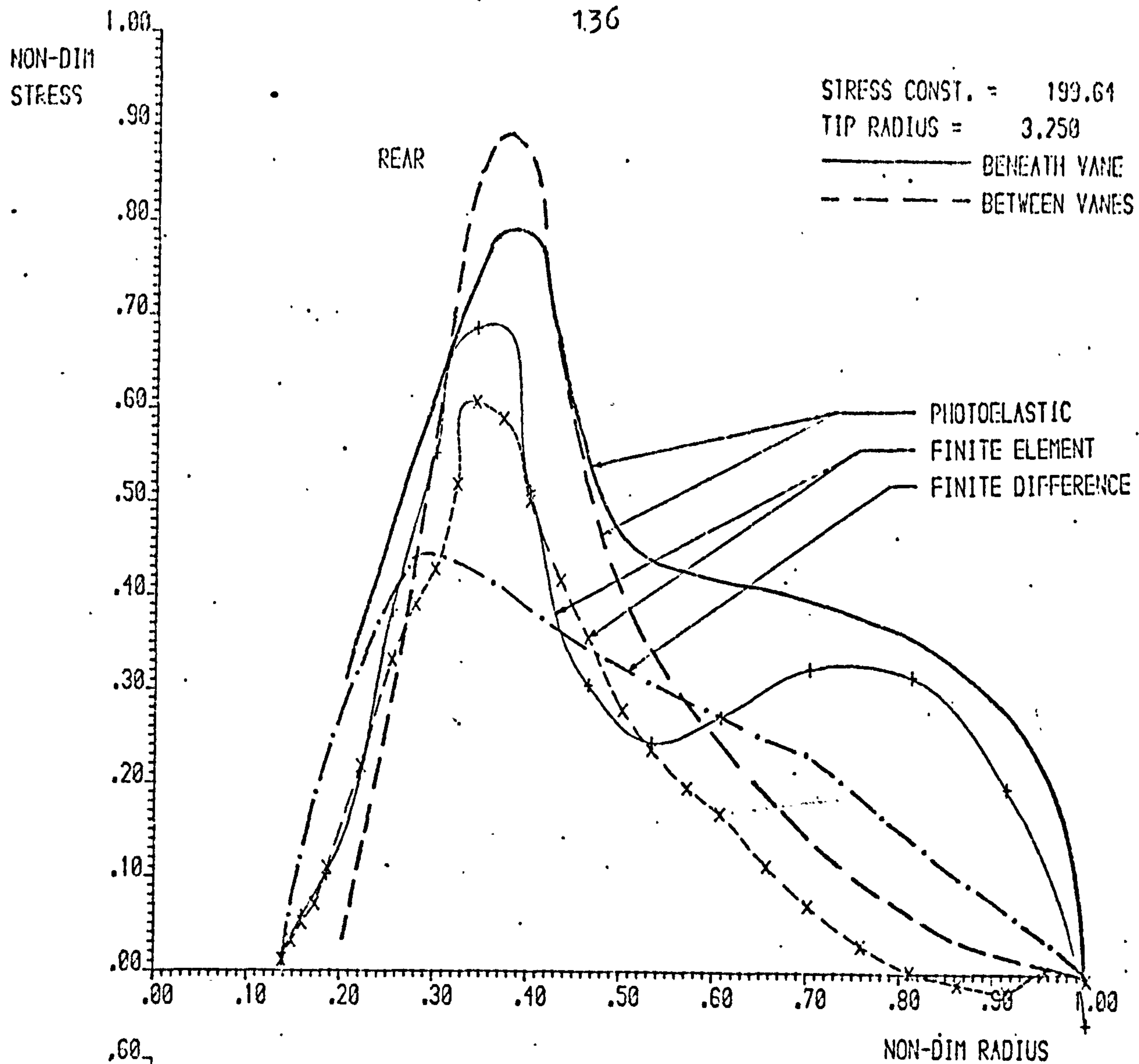
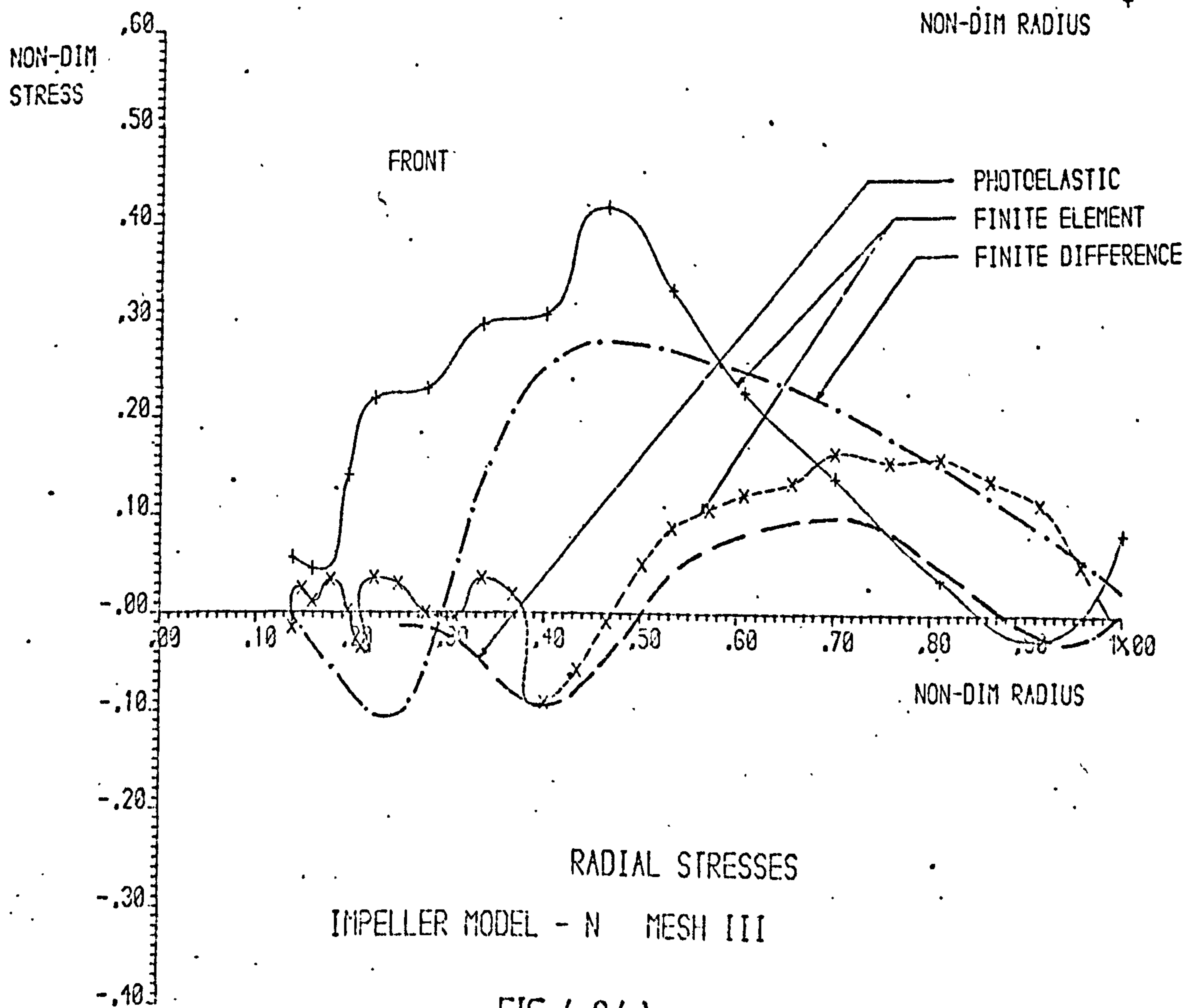
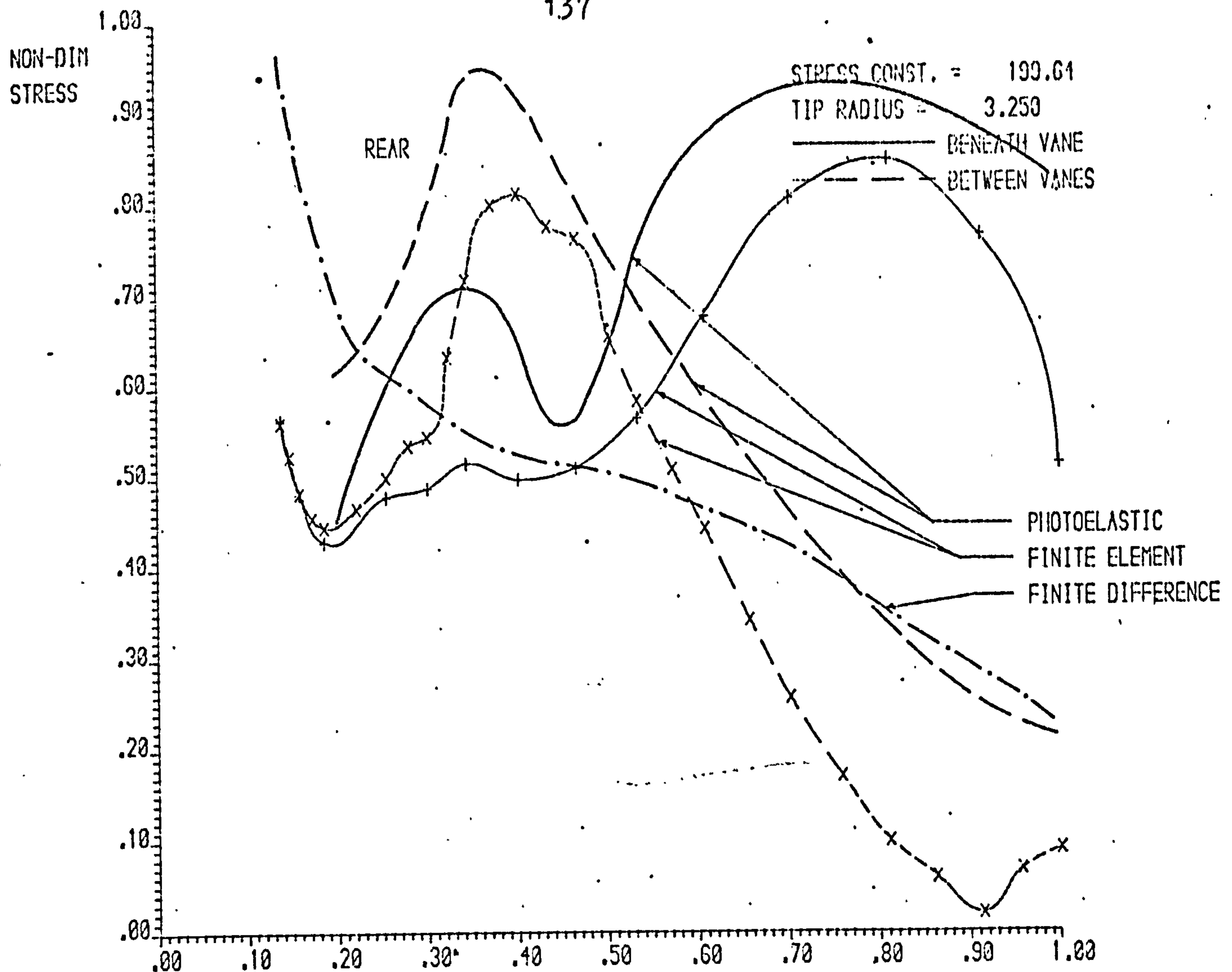
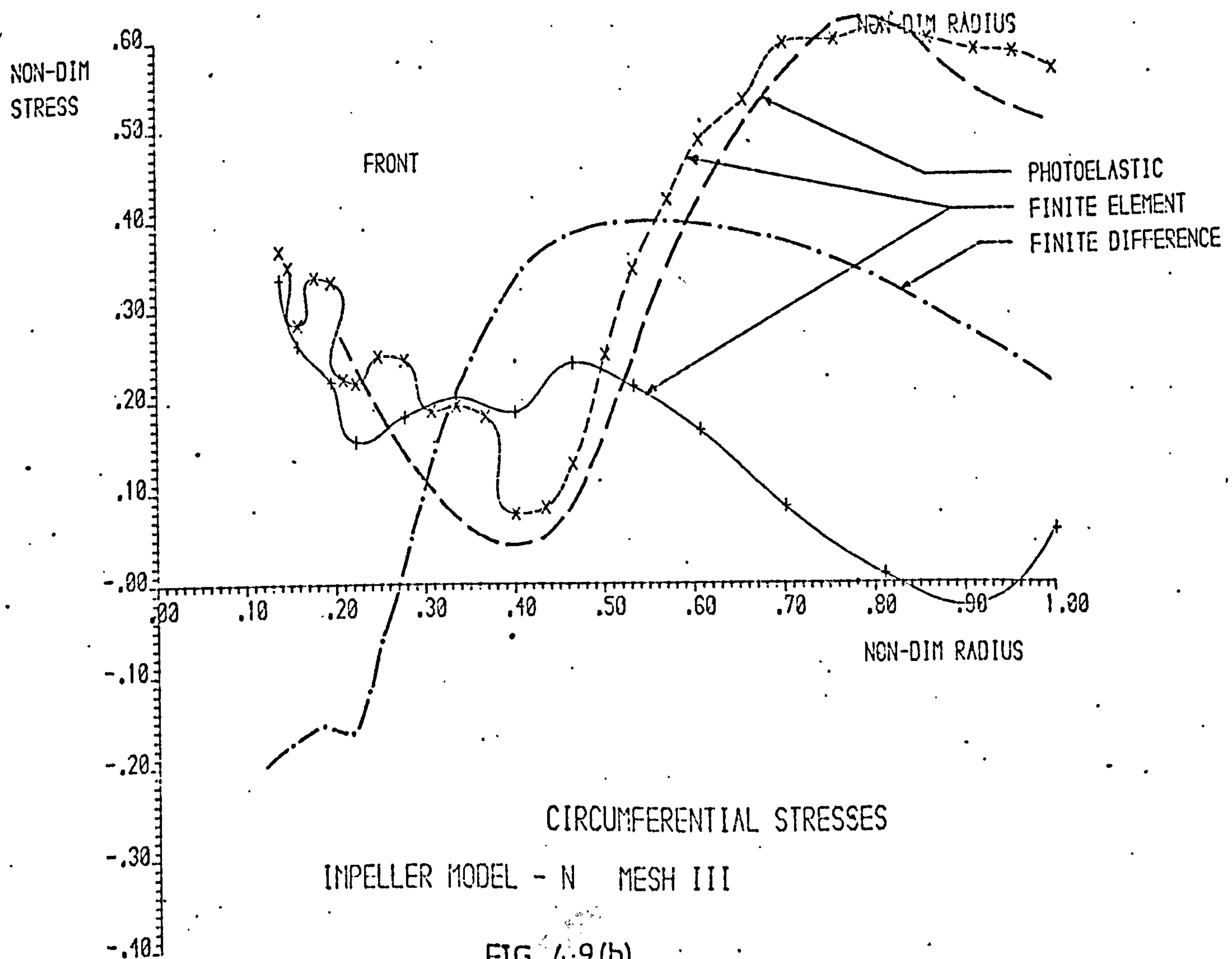


FIG 4-9(a)



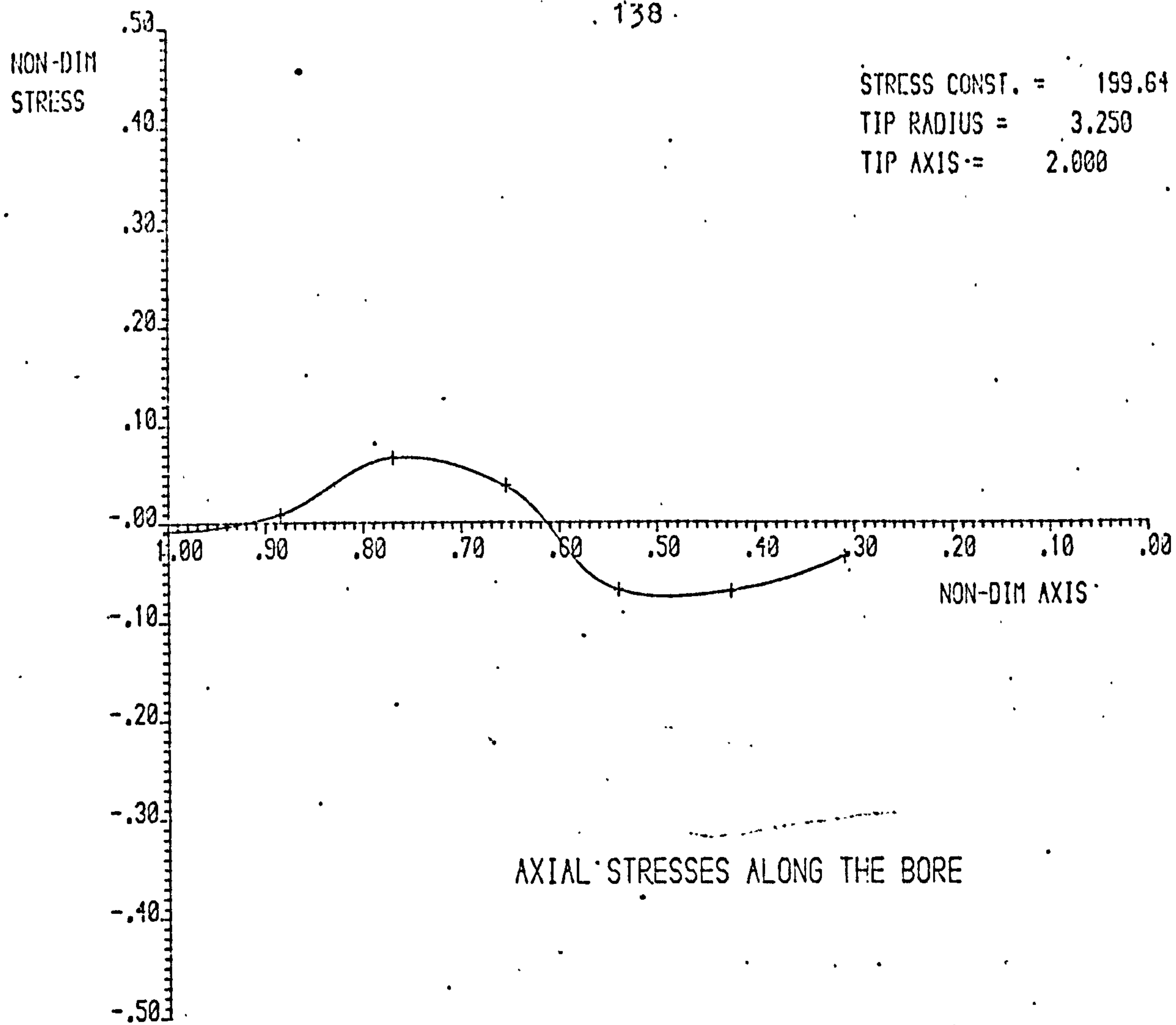
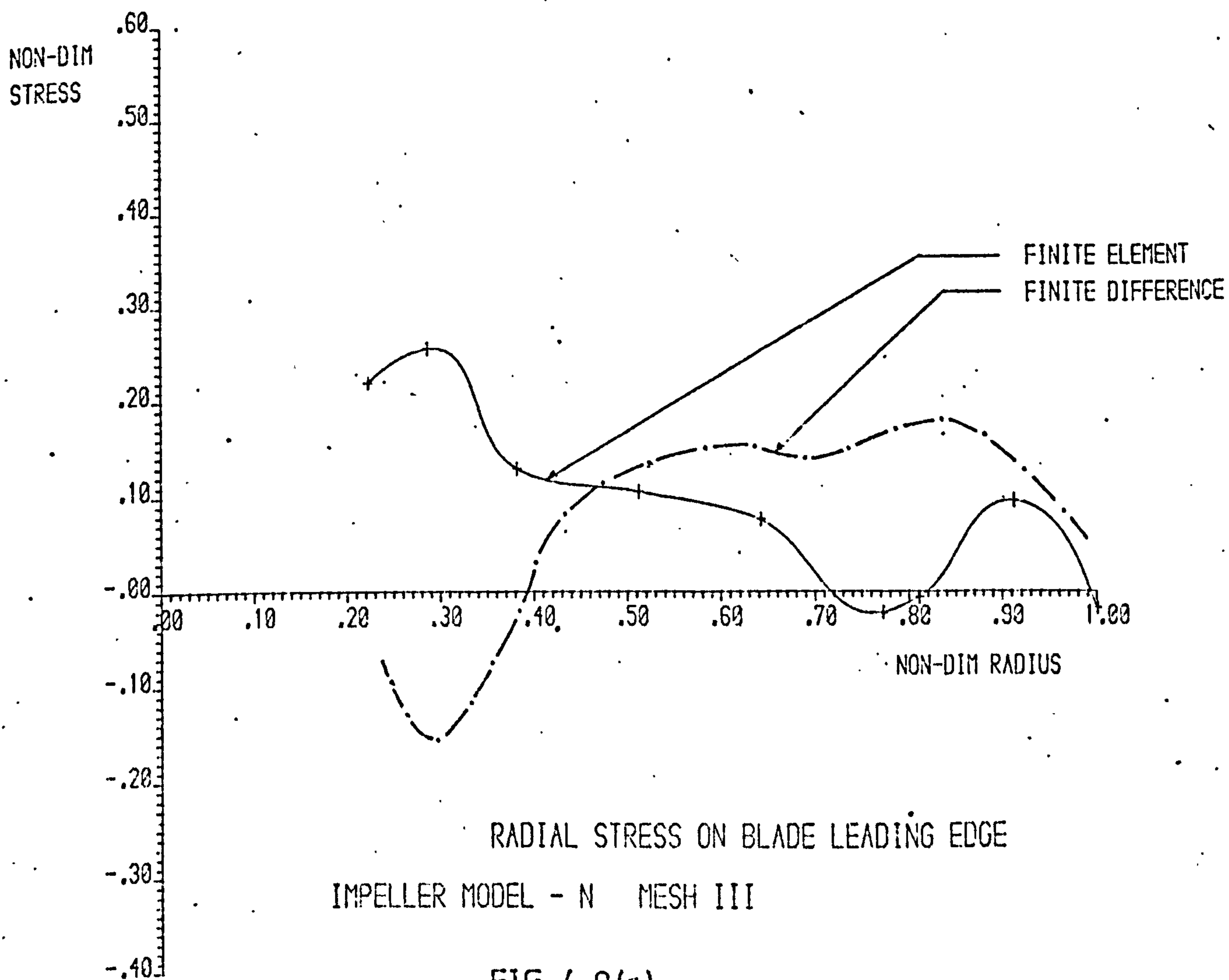
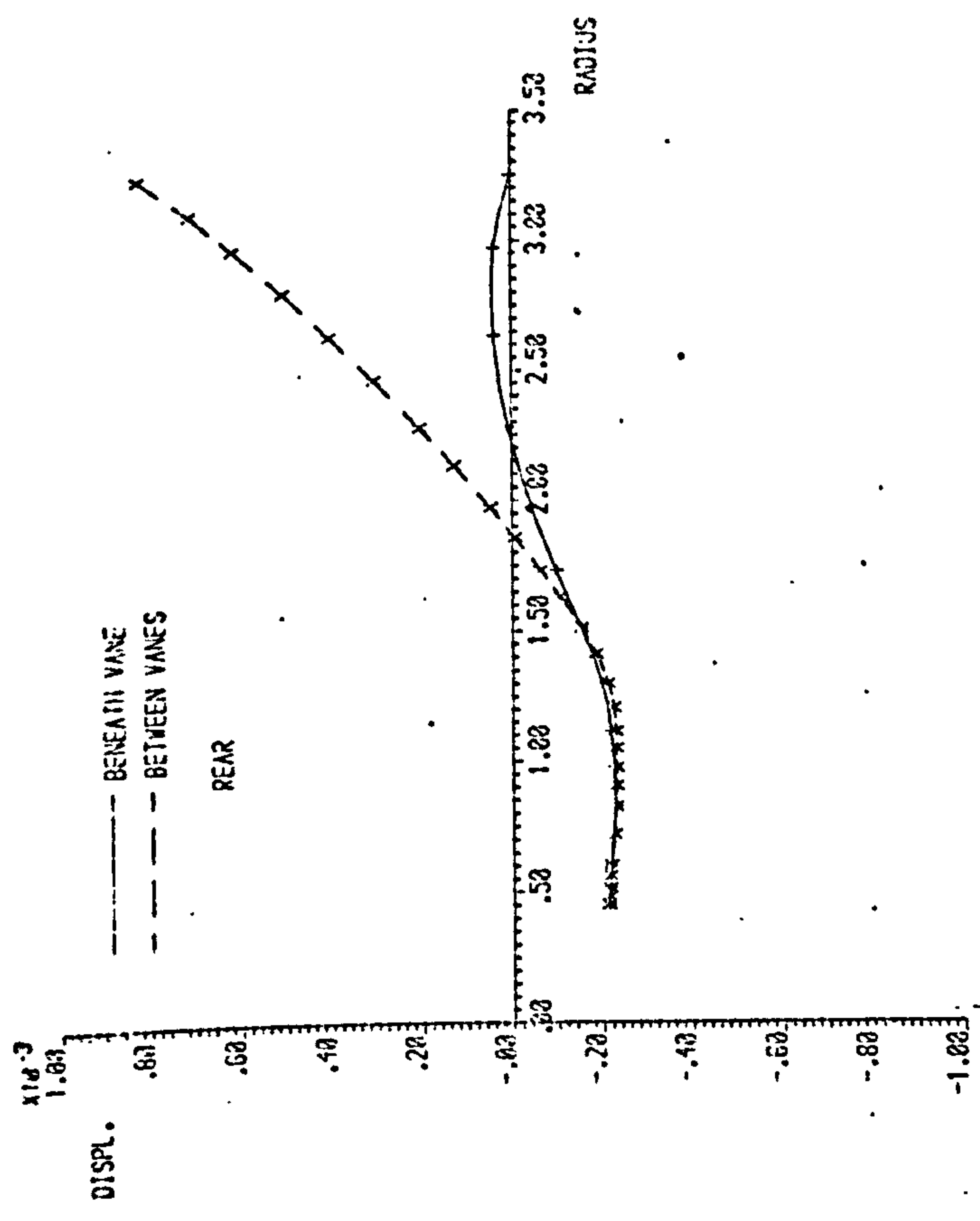
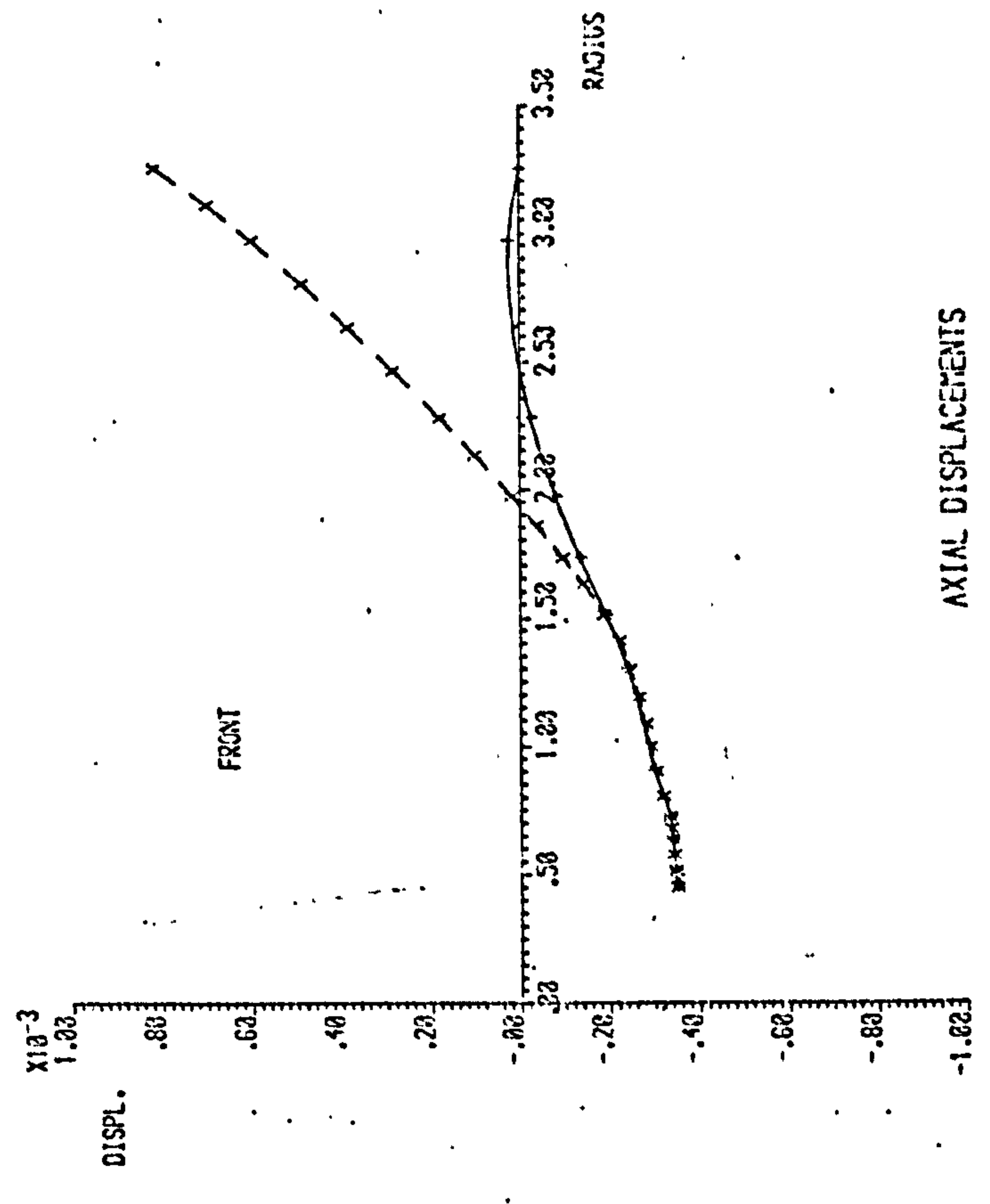
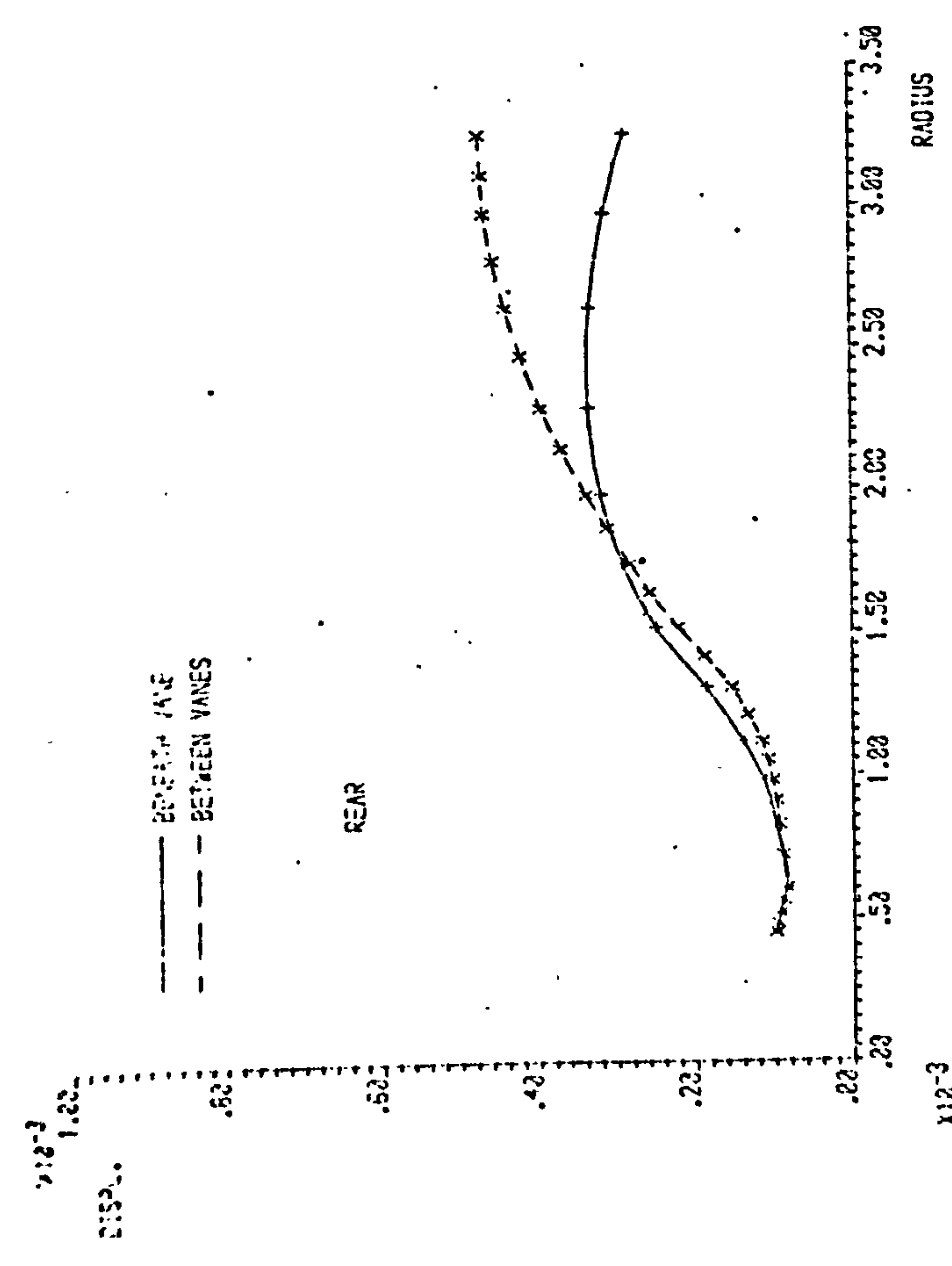
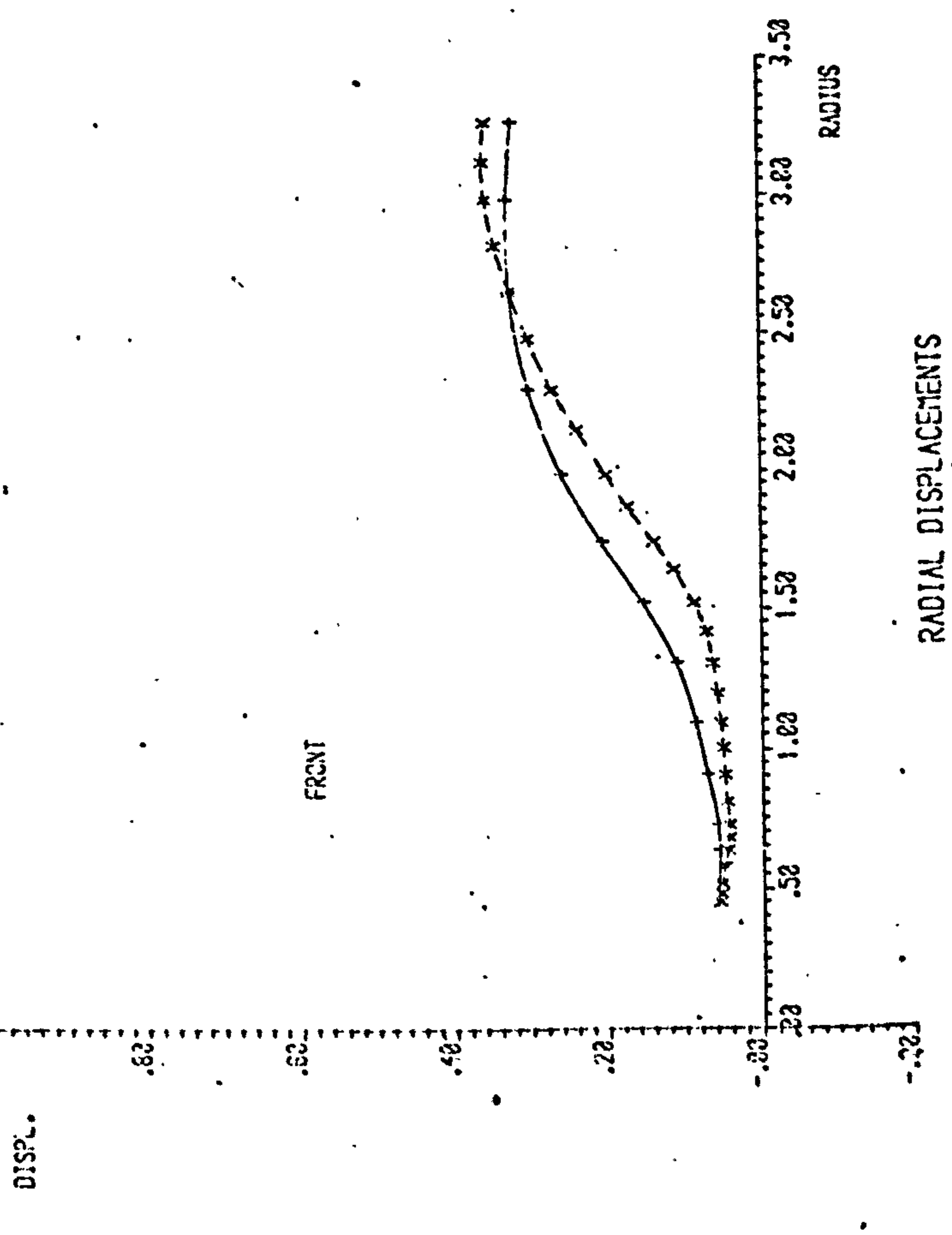


FIG 4-9(c)

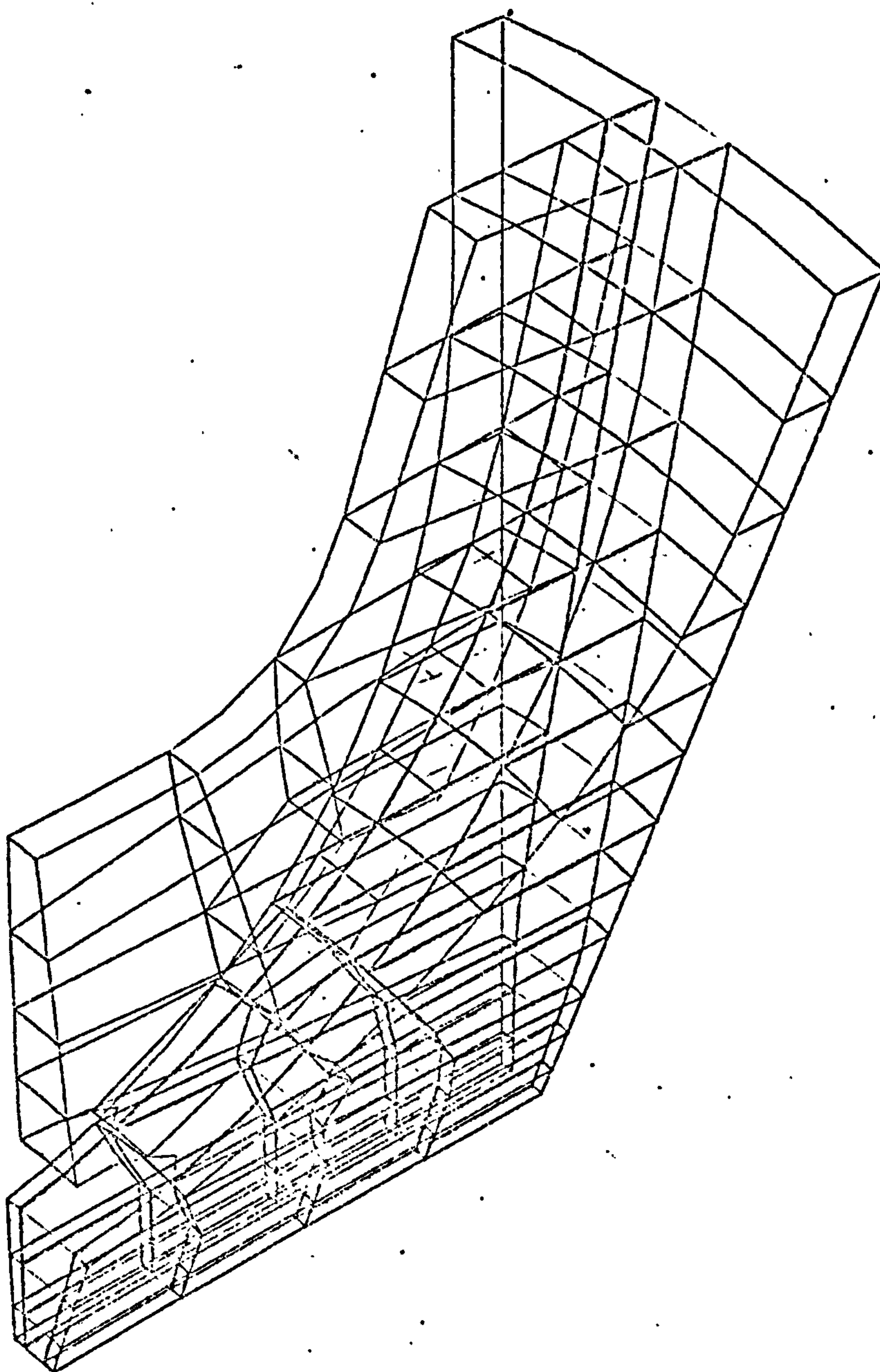


AXIAL DISPLACEMENTS
IMPELLER MODEL - N MESH III



RADIAL DISPLACEMENTS
IMPELLER MODEL - N MESH III

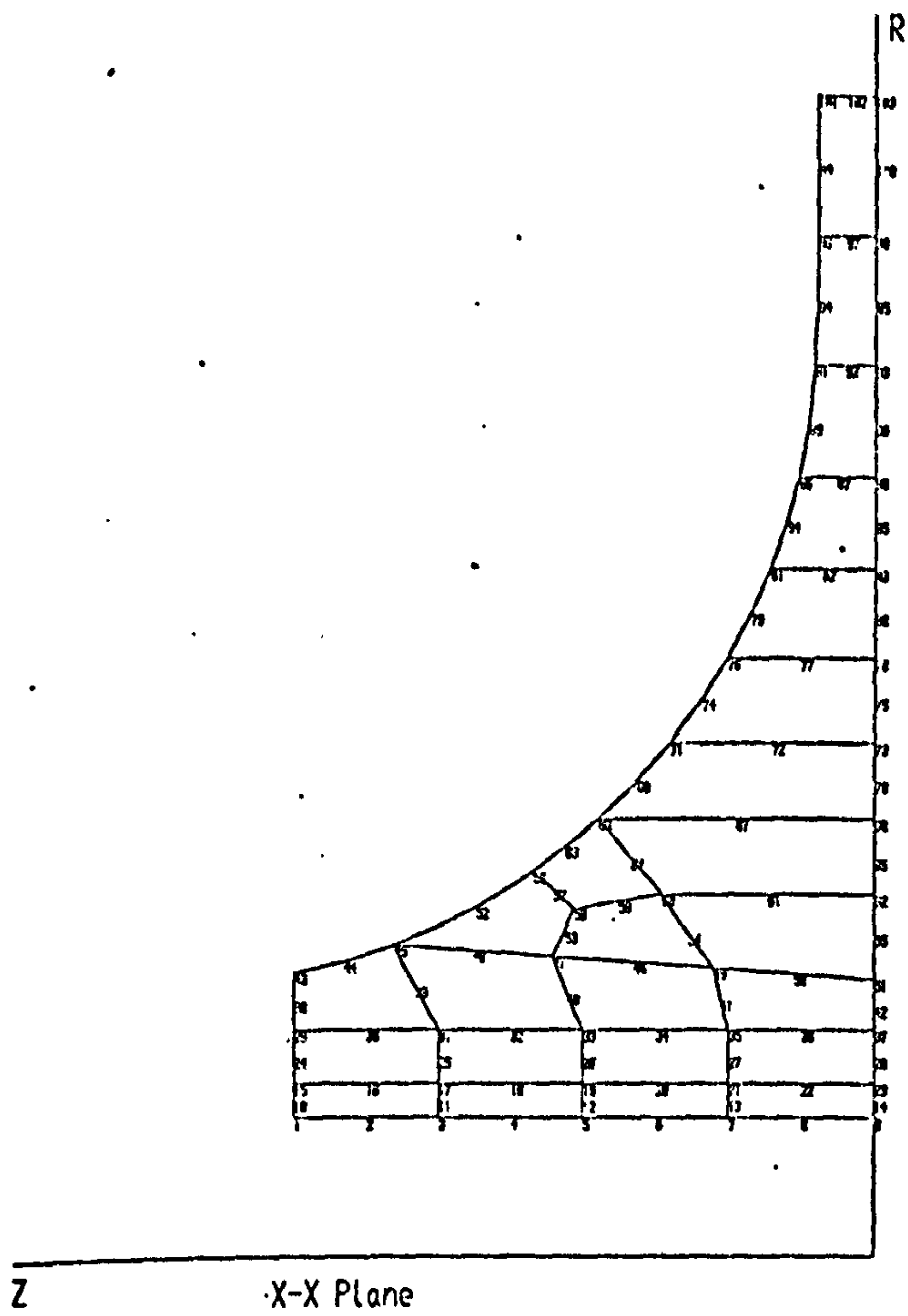
FIG 4-9(d)



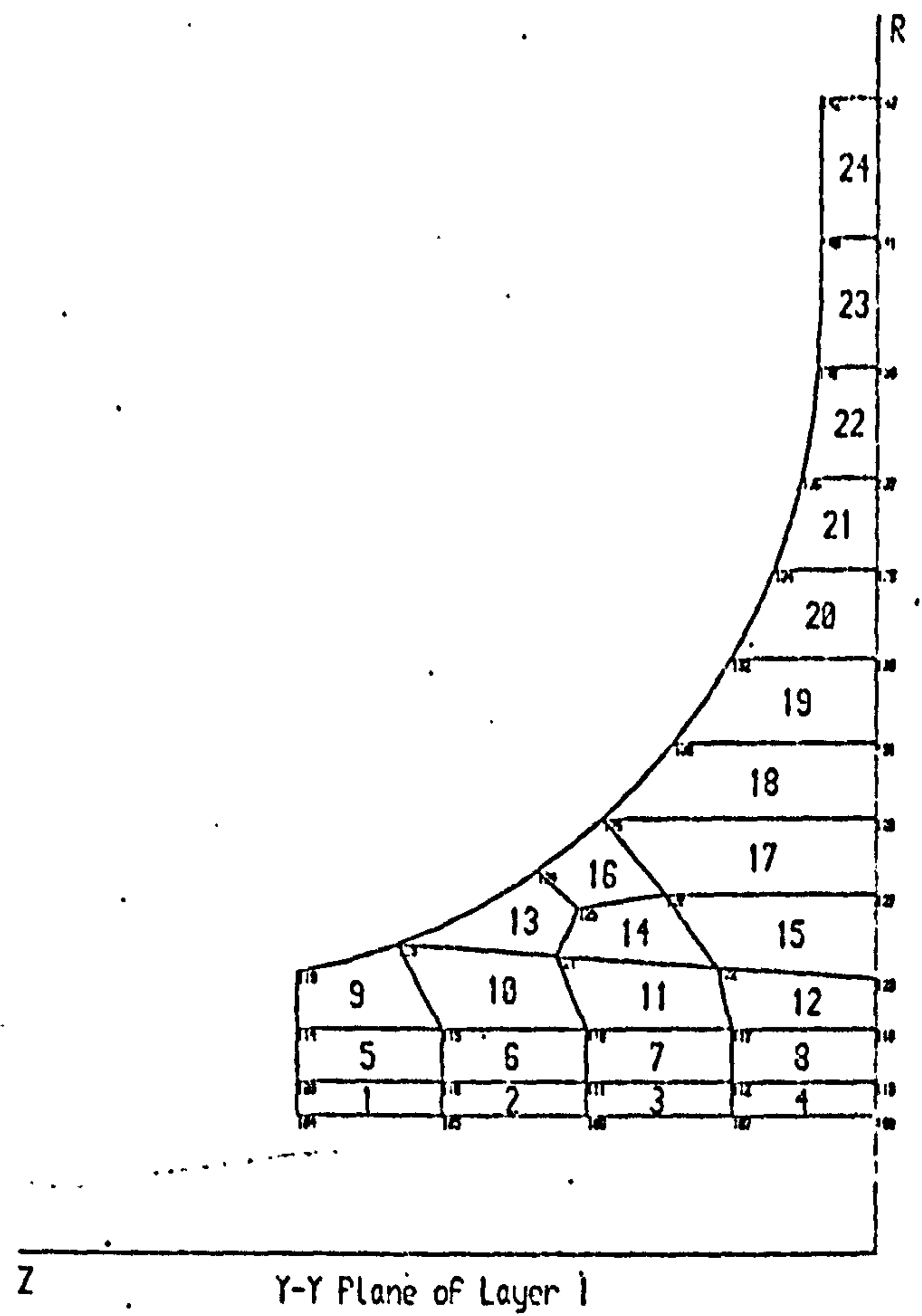
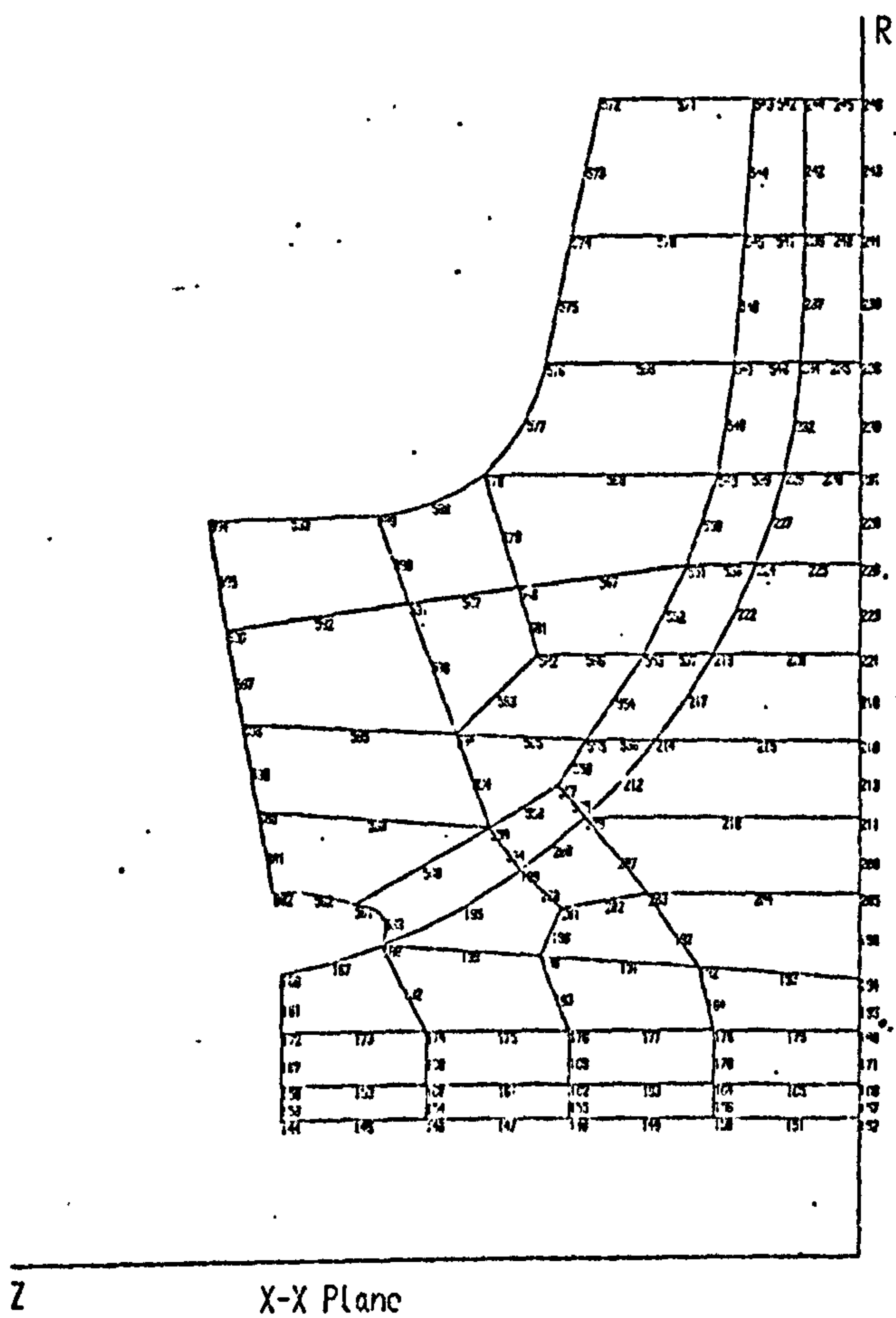
3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-R MESH III

FIG 4-10 (a)



IMPELLER MODEL-R MESH III

DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS
Sheet 1

IMPELLER MODEL-R MESH III

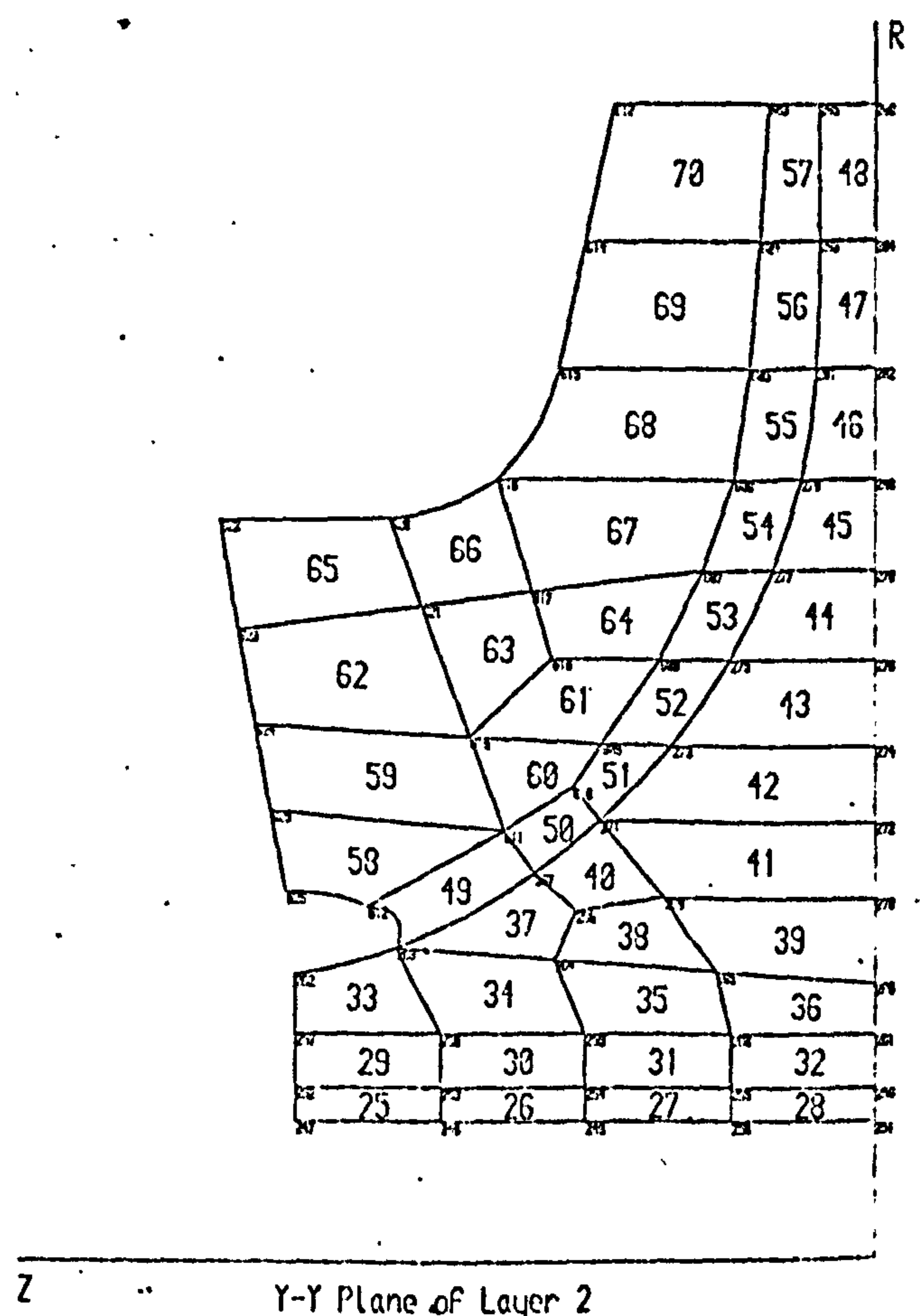
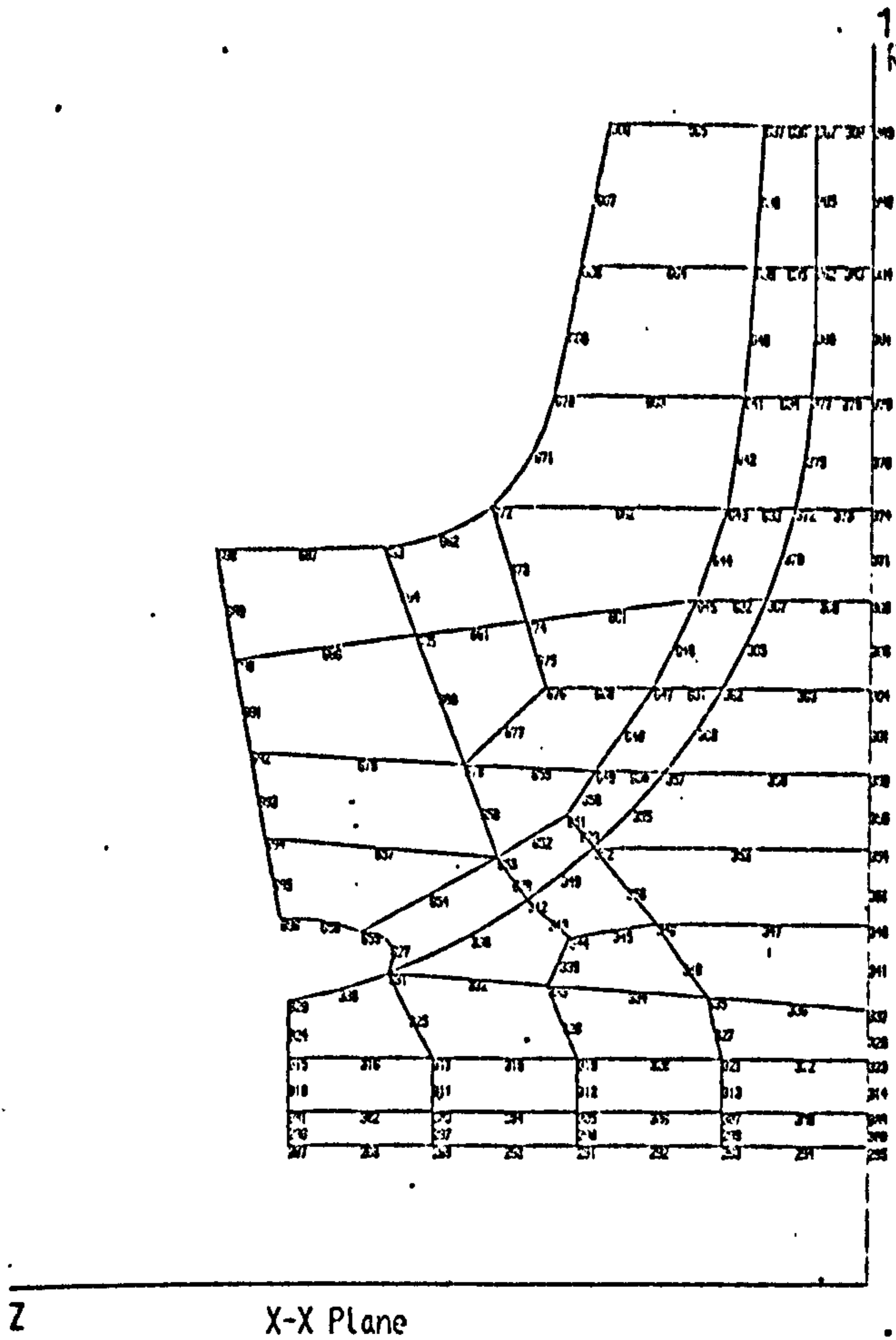
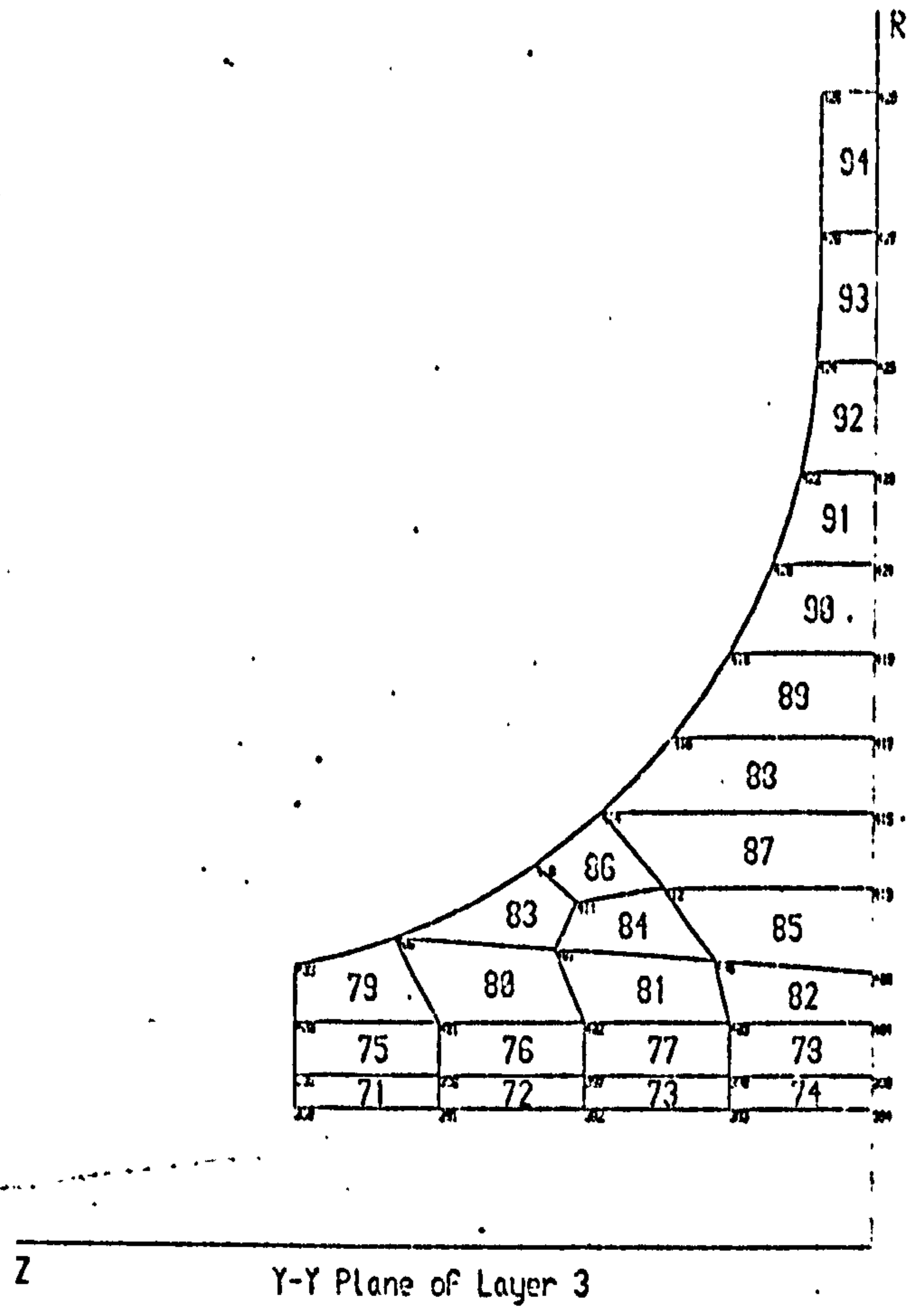
DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS
Sheet 2

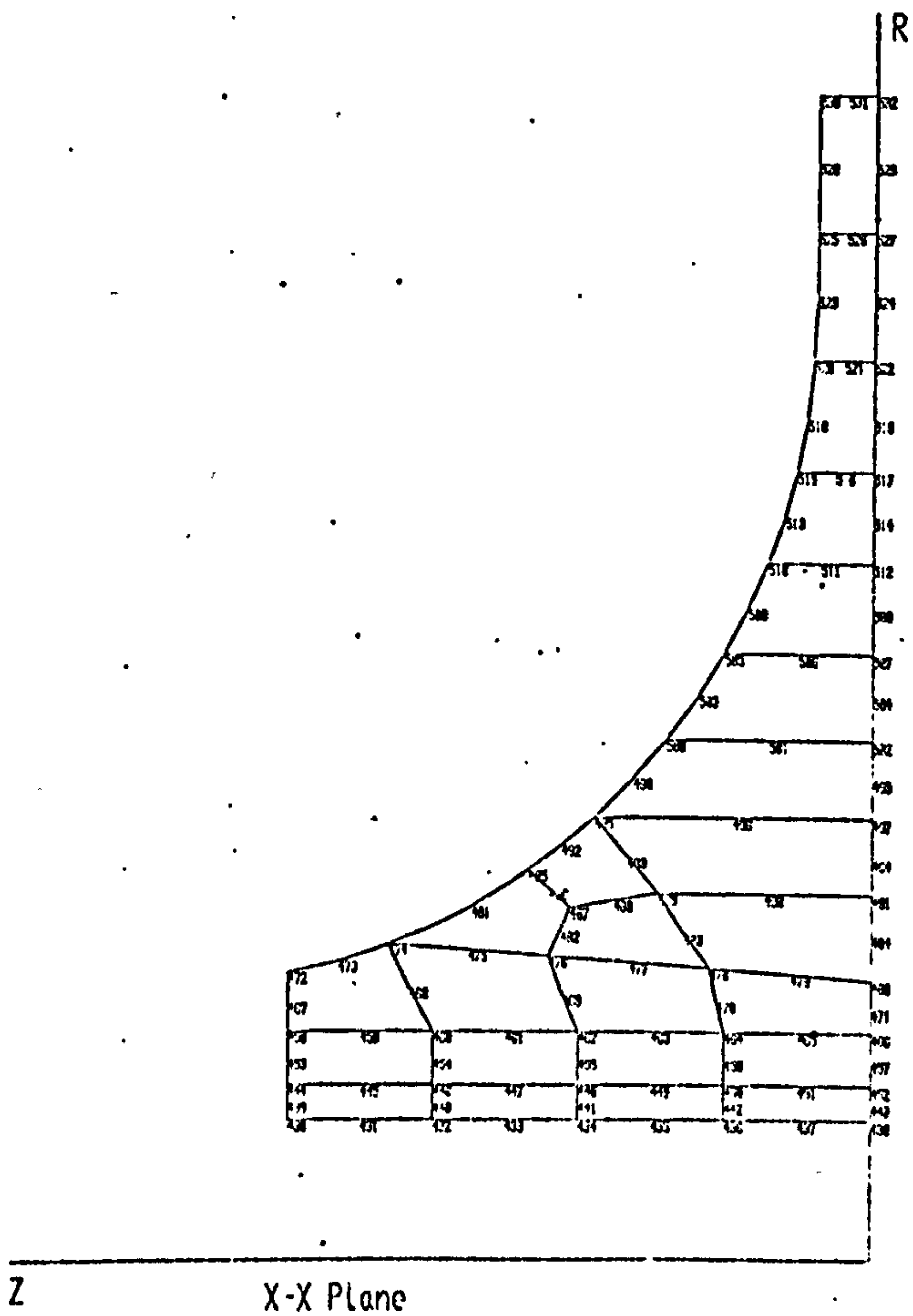
FIG 4.10(b)



IMPELLER MODEL-R MESH III

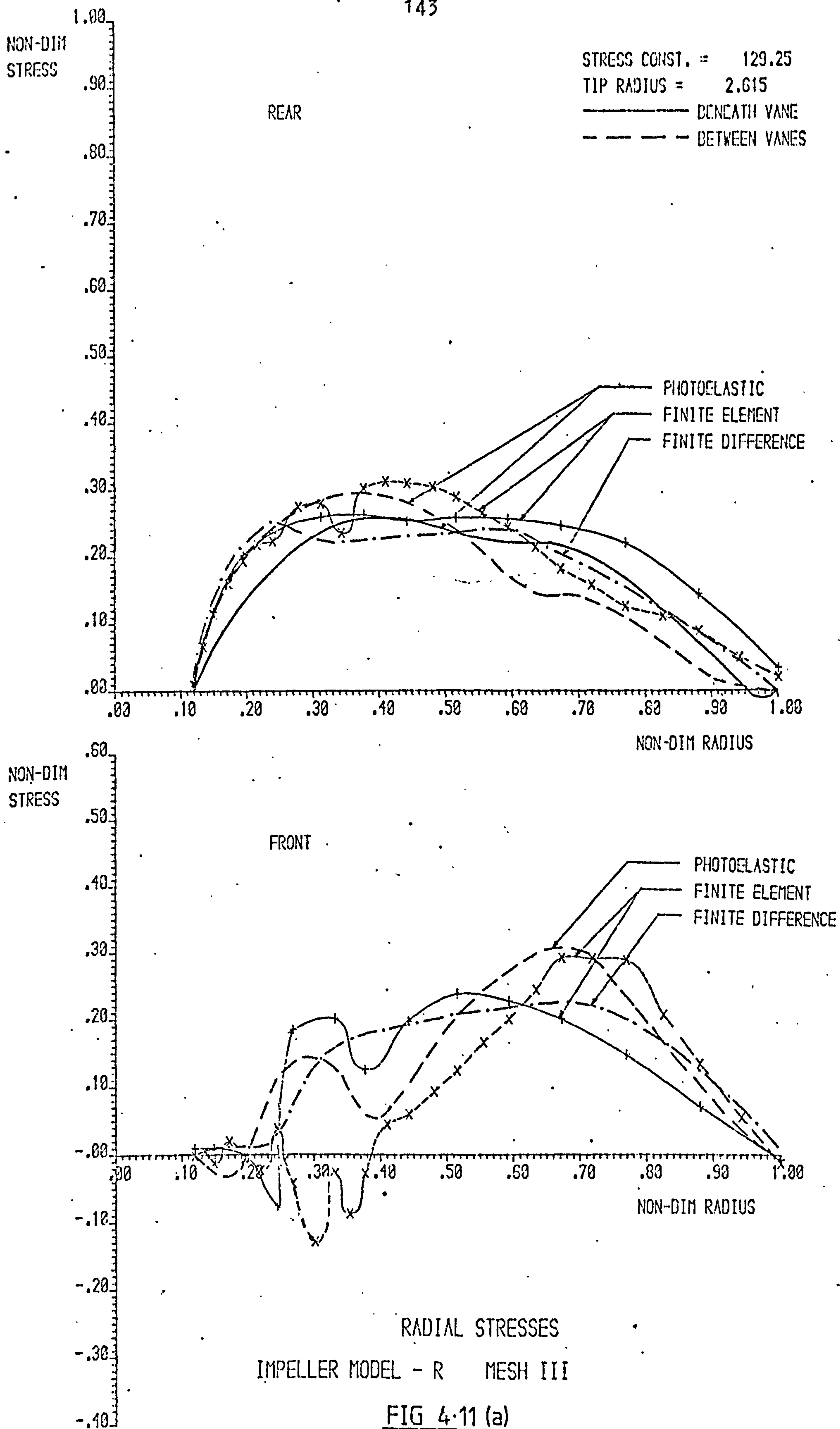


DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS
Sheet 3



IMPELLER MODEL-R MESH III

DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS
Sheet 4



NON-DIM
STRESS

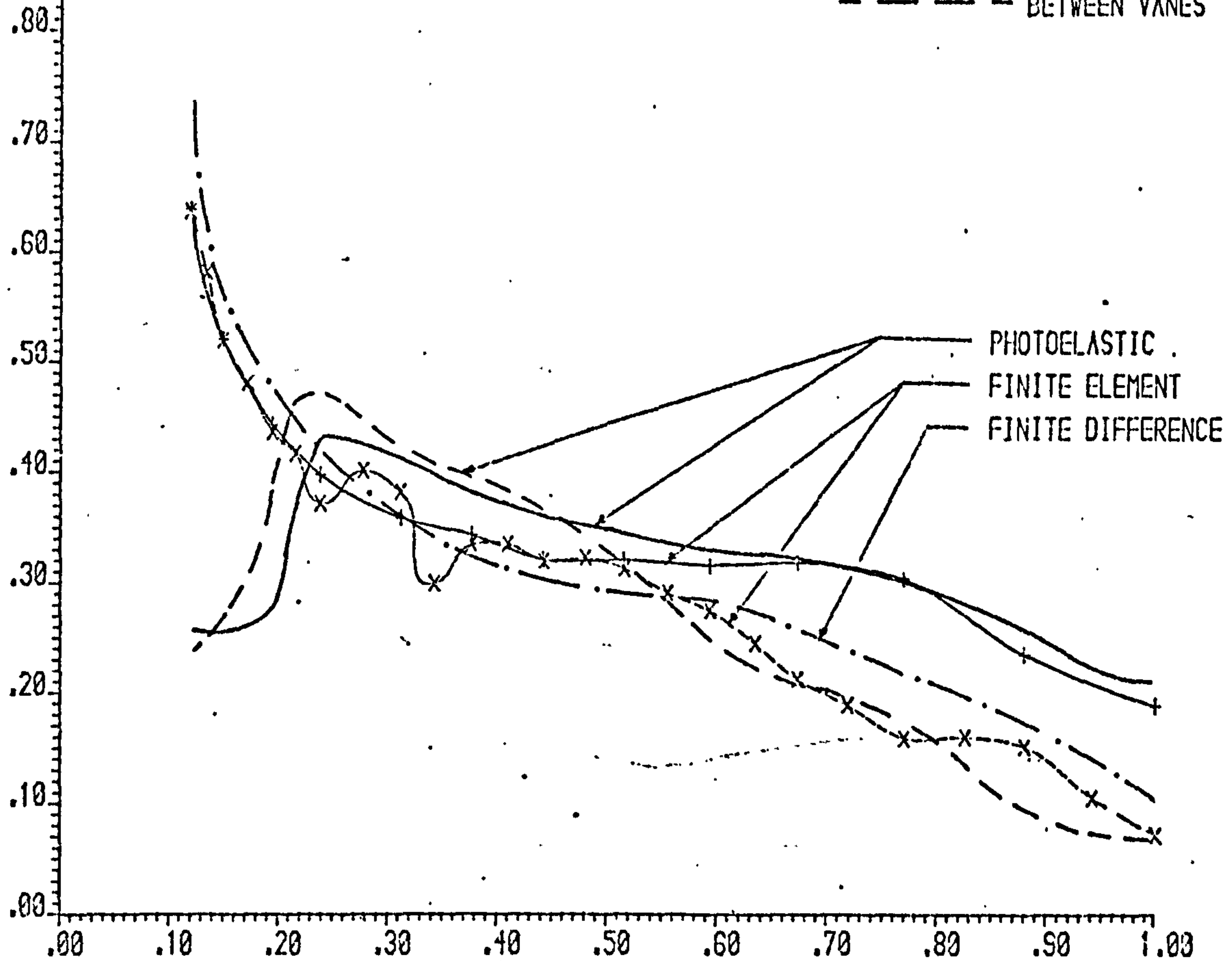
STRESS CONST. = 129.25

TIP RADIUS = 2.615

REAR

BENEATH VANE

BETWEEN VANES



NON-DIM RADIUS

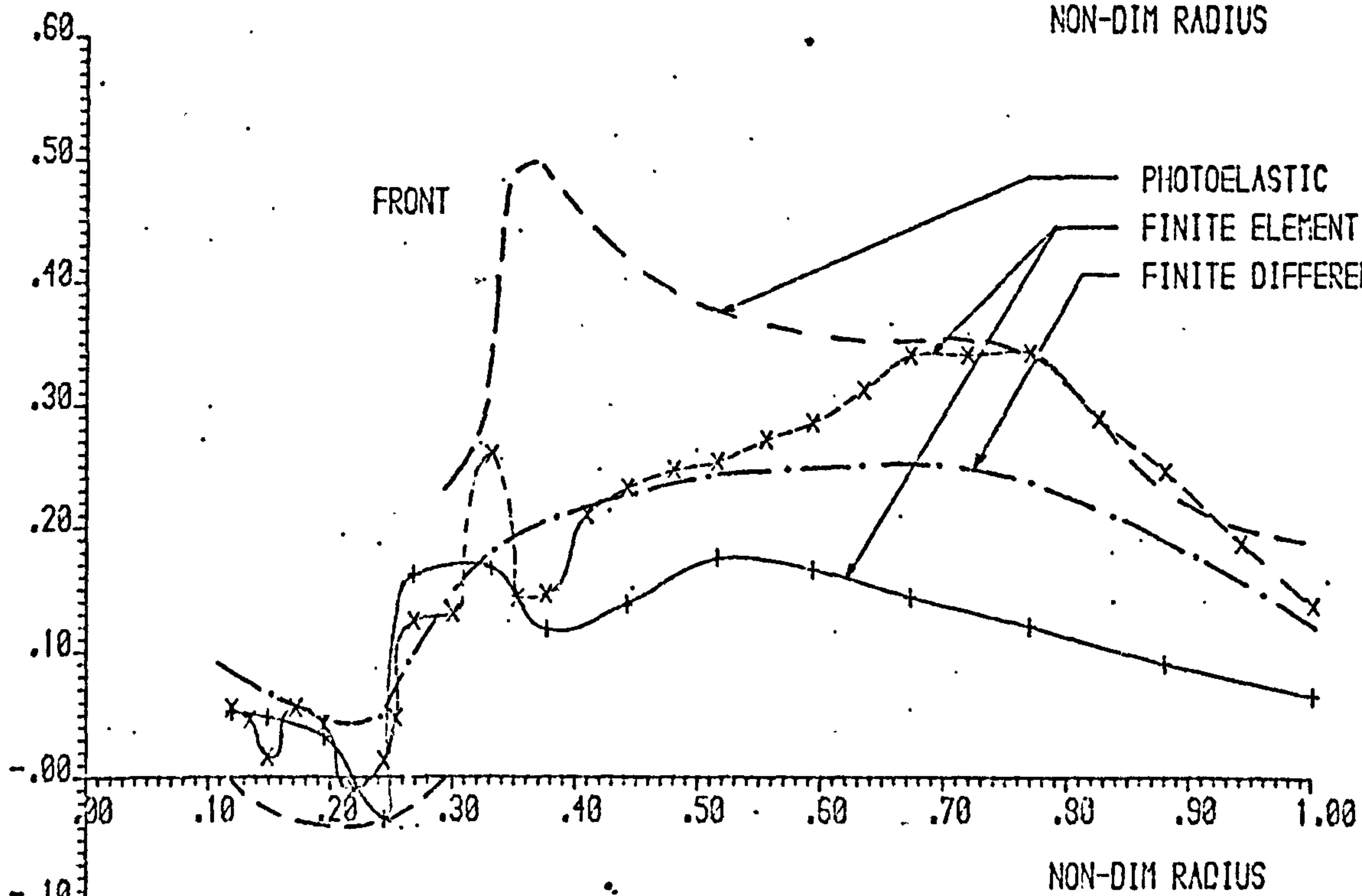
NON-DIM
STRESS

FRONT

PHOTOELASTIC

FINITE ELEMENT

FINITE DIFFERENCE



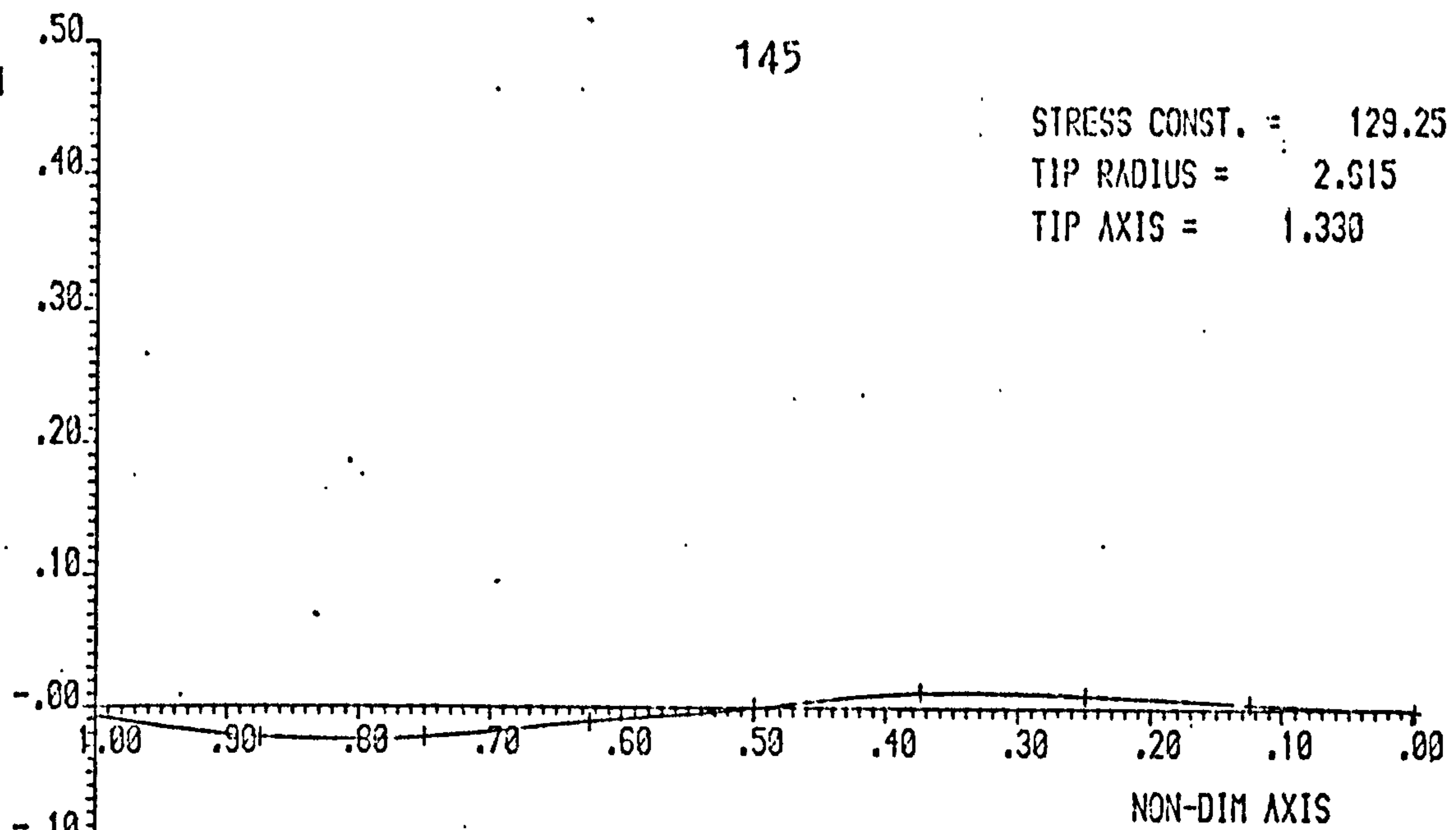
CIRCUMFERENTIAL STRESSES

IMPELLER MODEL - R MESH III

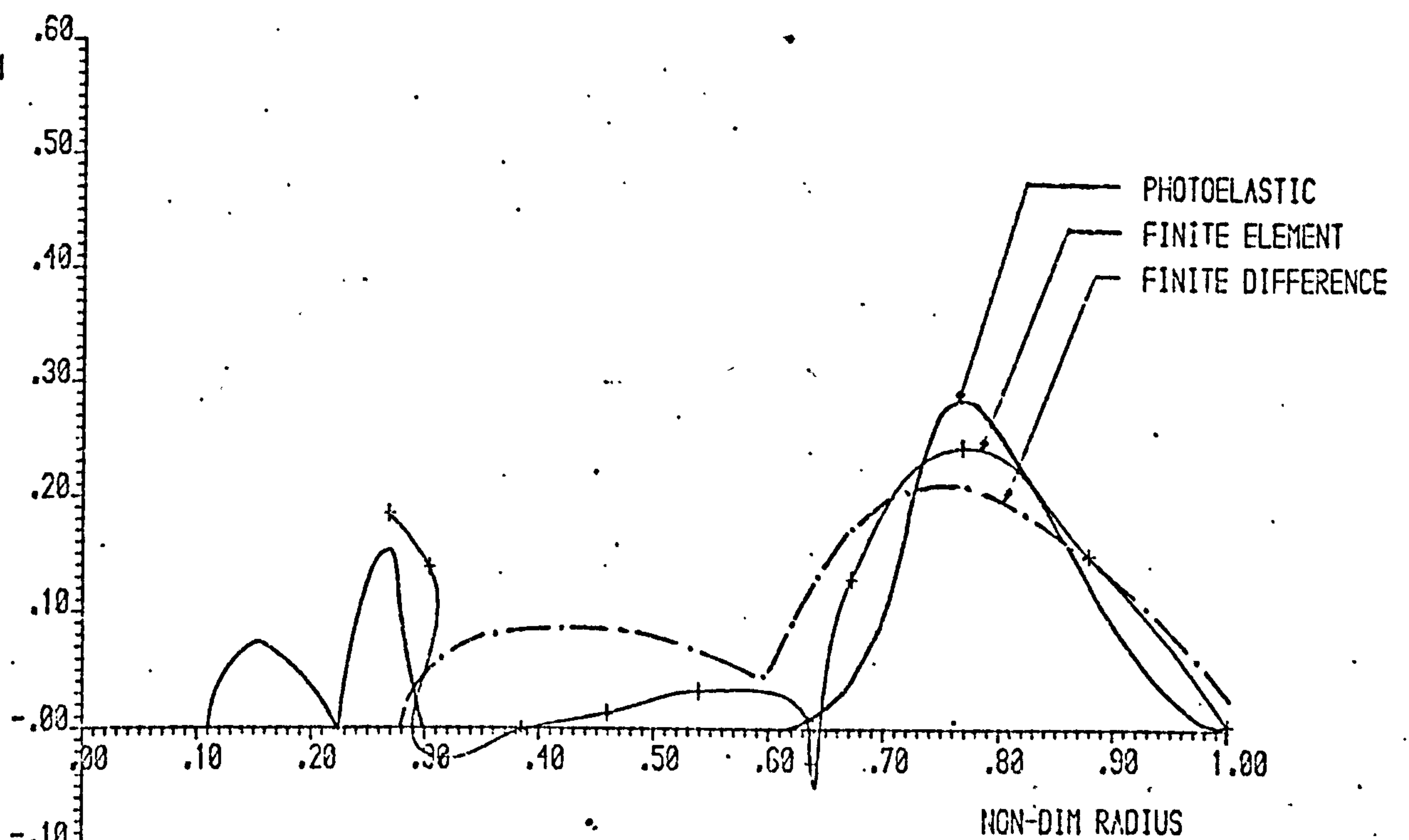
FIG 4-11(b)

NON-DIM
STRESS

STRESS CONST. = 129.25
 TIP RADIUS = 2.615
 TIP AXIS = 1.330



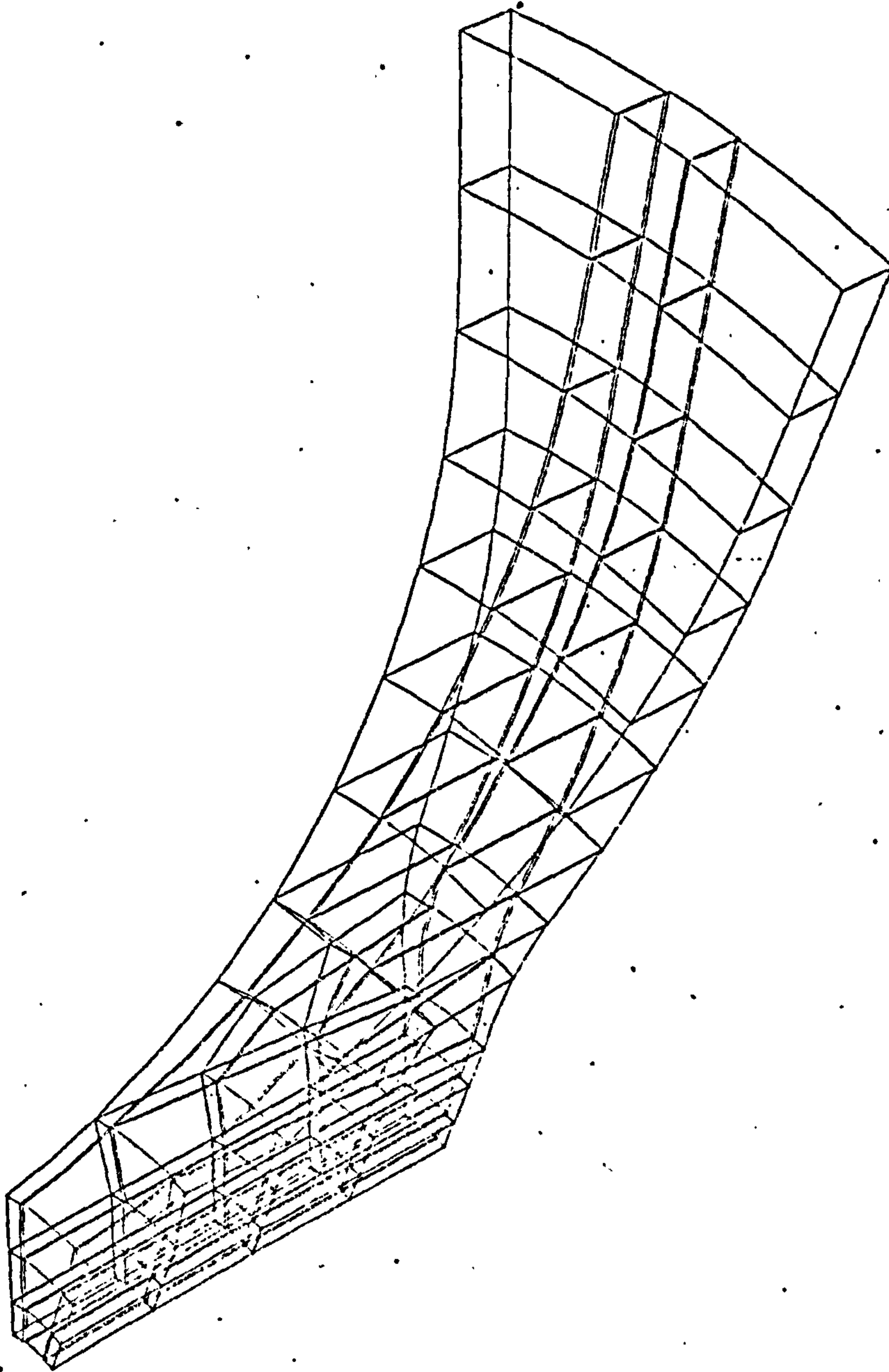
AXIAL STRESSES ALONG THE BORE

NON-DIM
STRESS

RADIAL STRESS ON BLADE LEADING EDGE

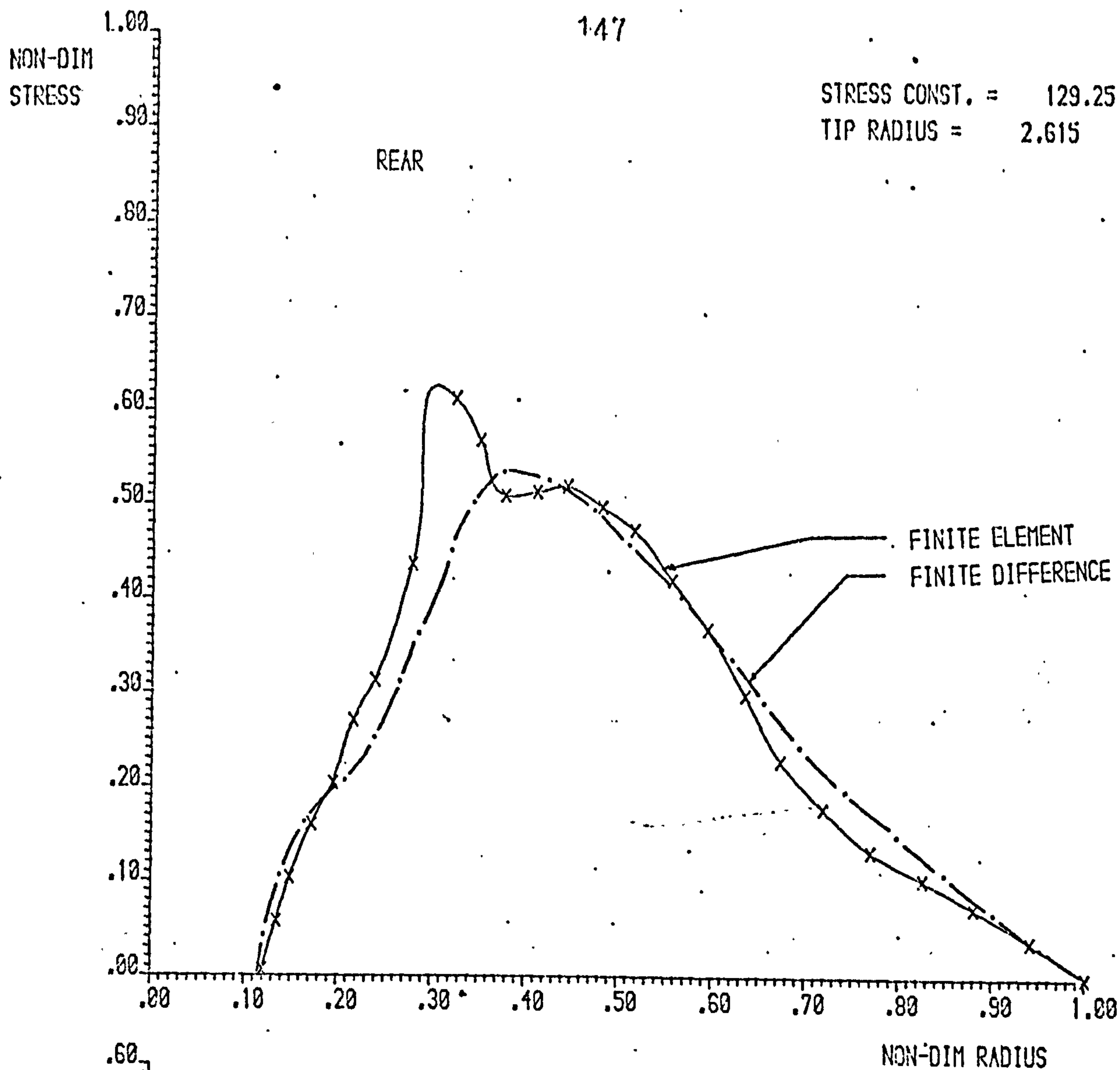
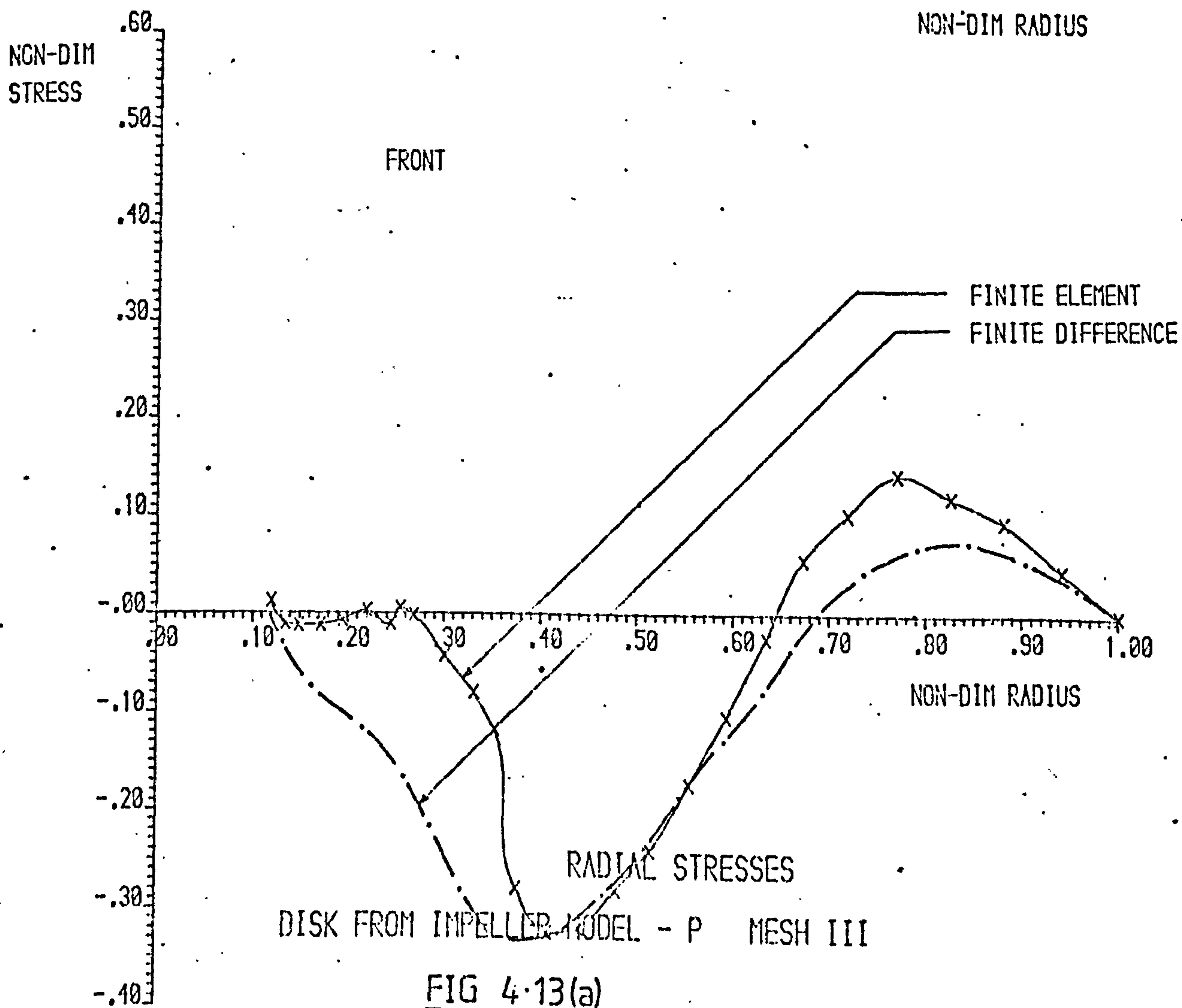
IMPELLER MODEL - R MESH III

FIG 4-11(c)



3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS

FIG 4.12 DISK FROM IMPELLER MODEL-P MESH III



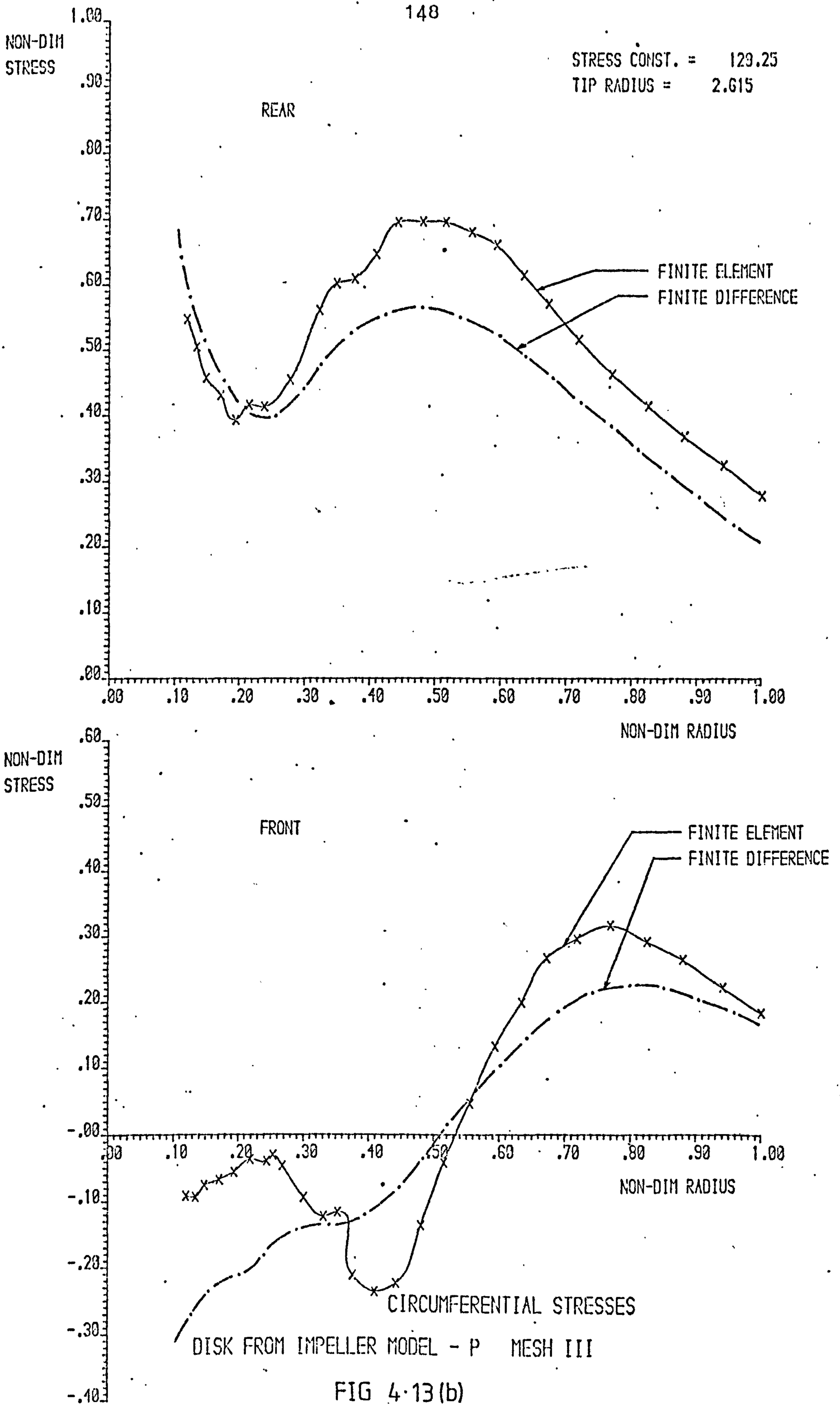


FIG 4-13(b)

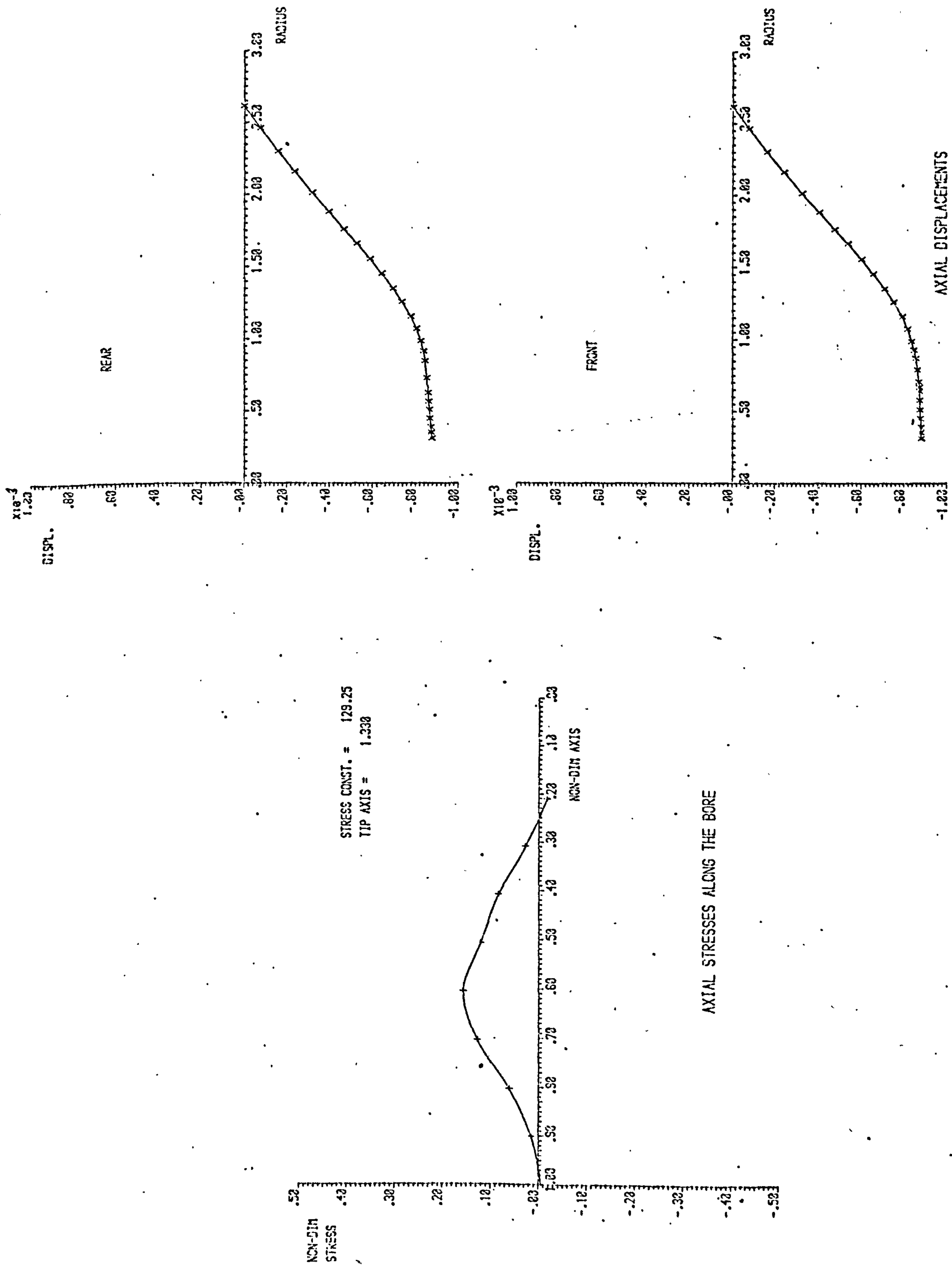
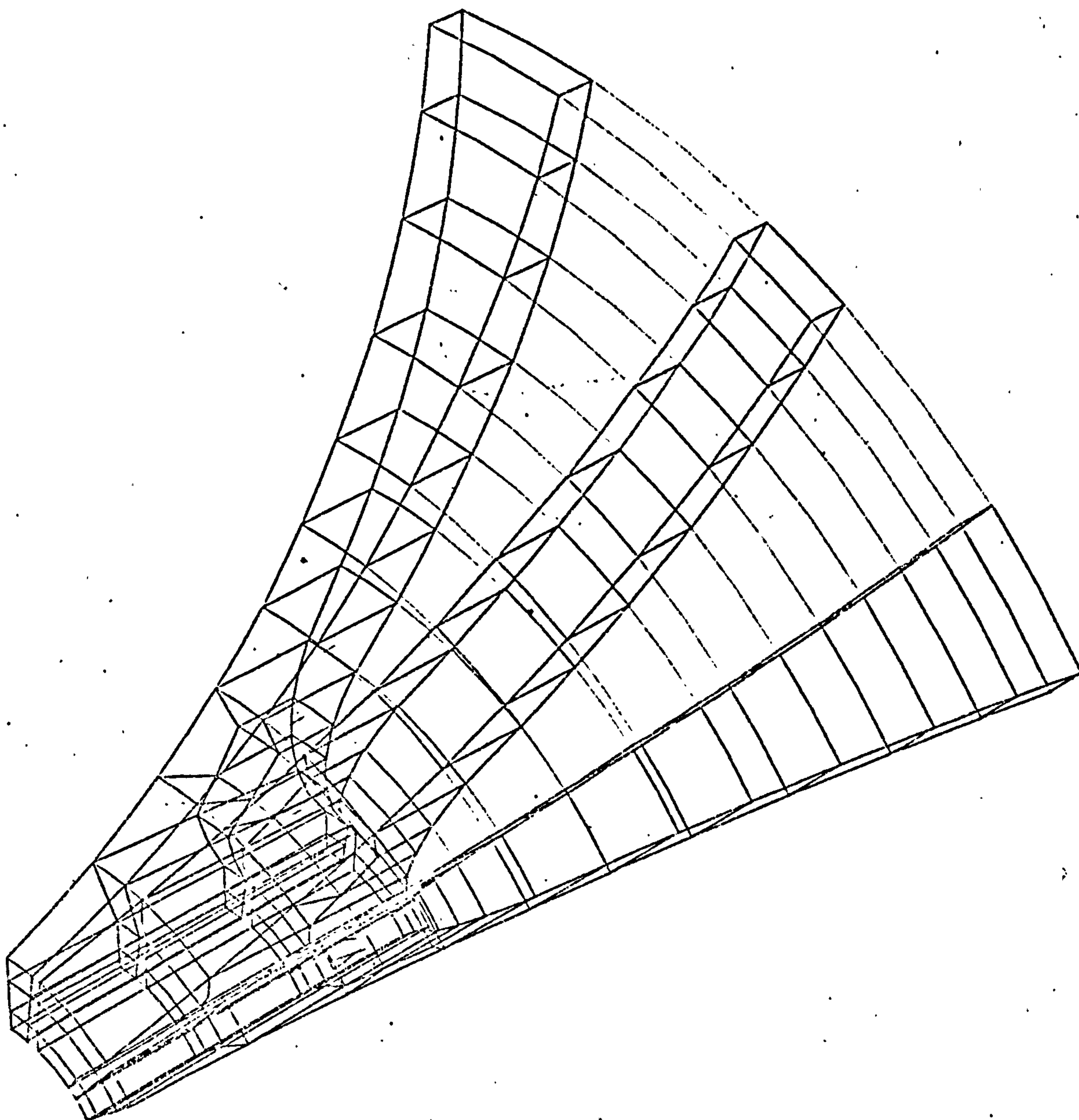


FIG 4.13(c)



3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN MM SCALE: 40.00 TIMES THE SUPPLIED UNITS

FIG 4-14 DISK FROM IMPELLER MODEL-N MESH III

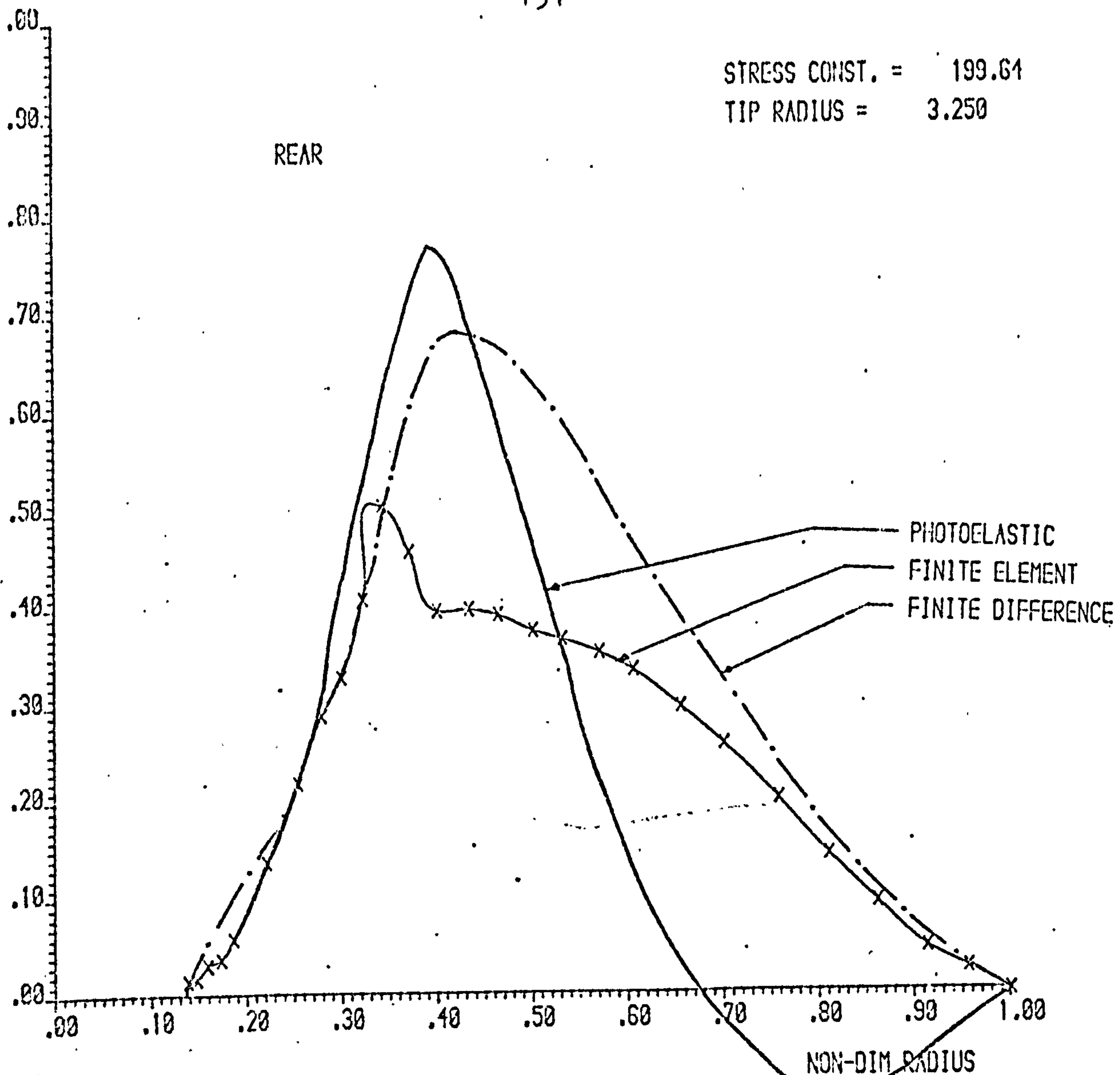
Text cut off in original

NON-DIM
STRESS

STRESS CONST. = 199.64

TIP RADIUS = 3.250

REAR

NON-DIM
STRESS

FRONT

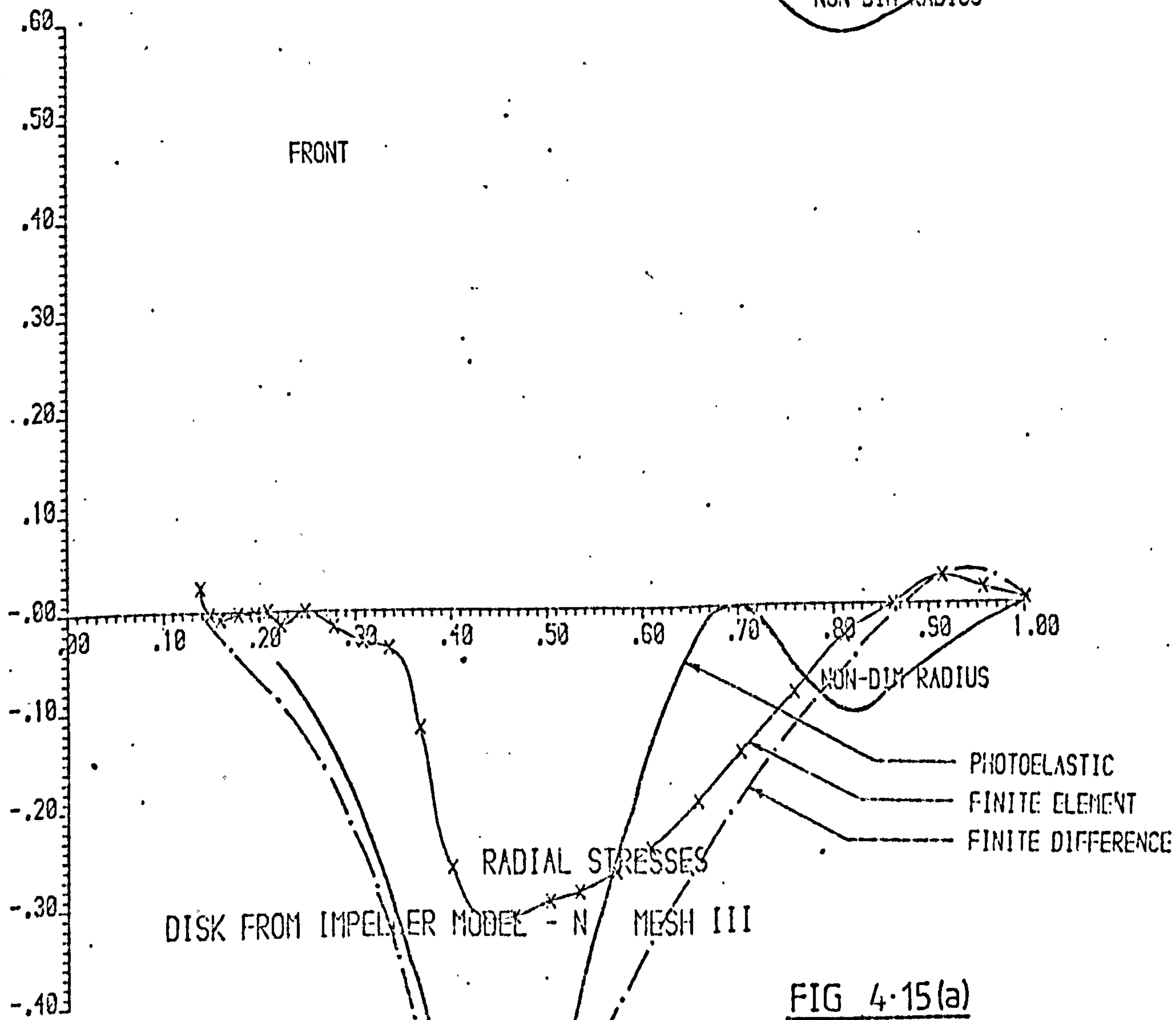
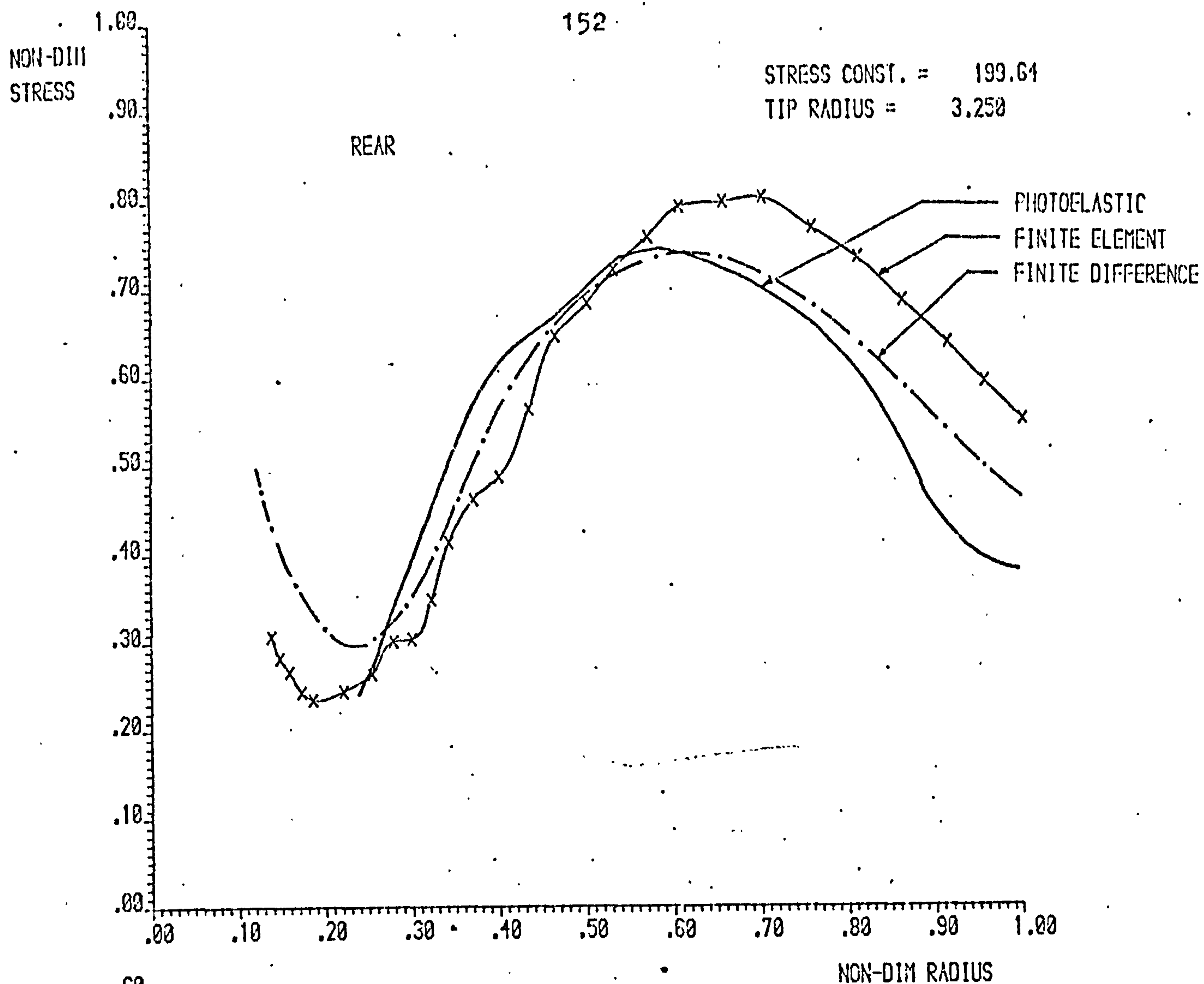
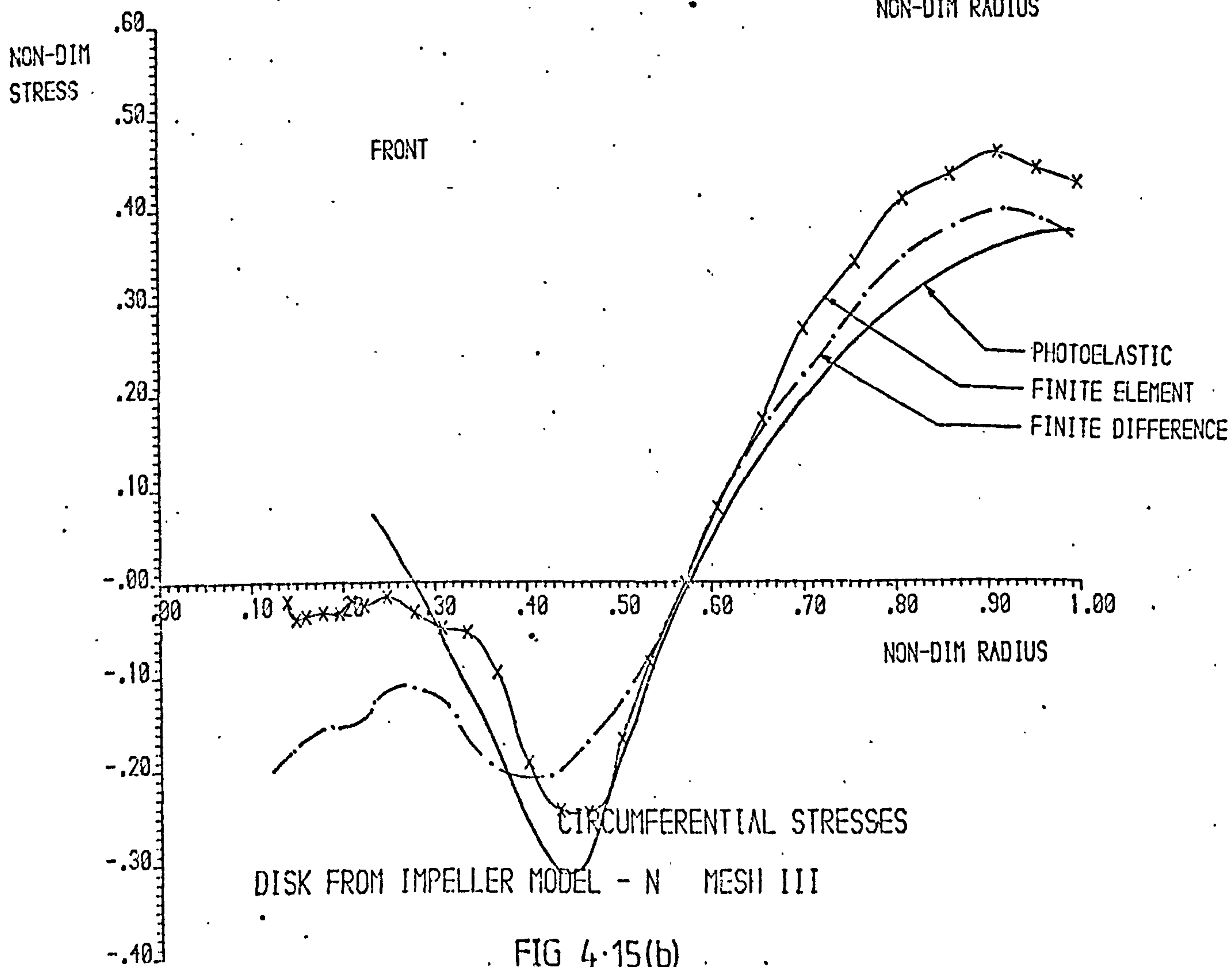
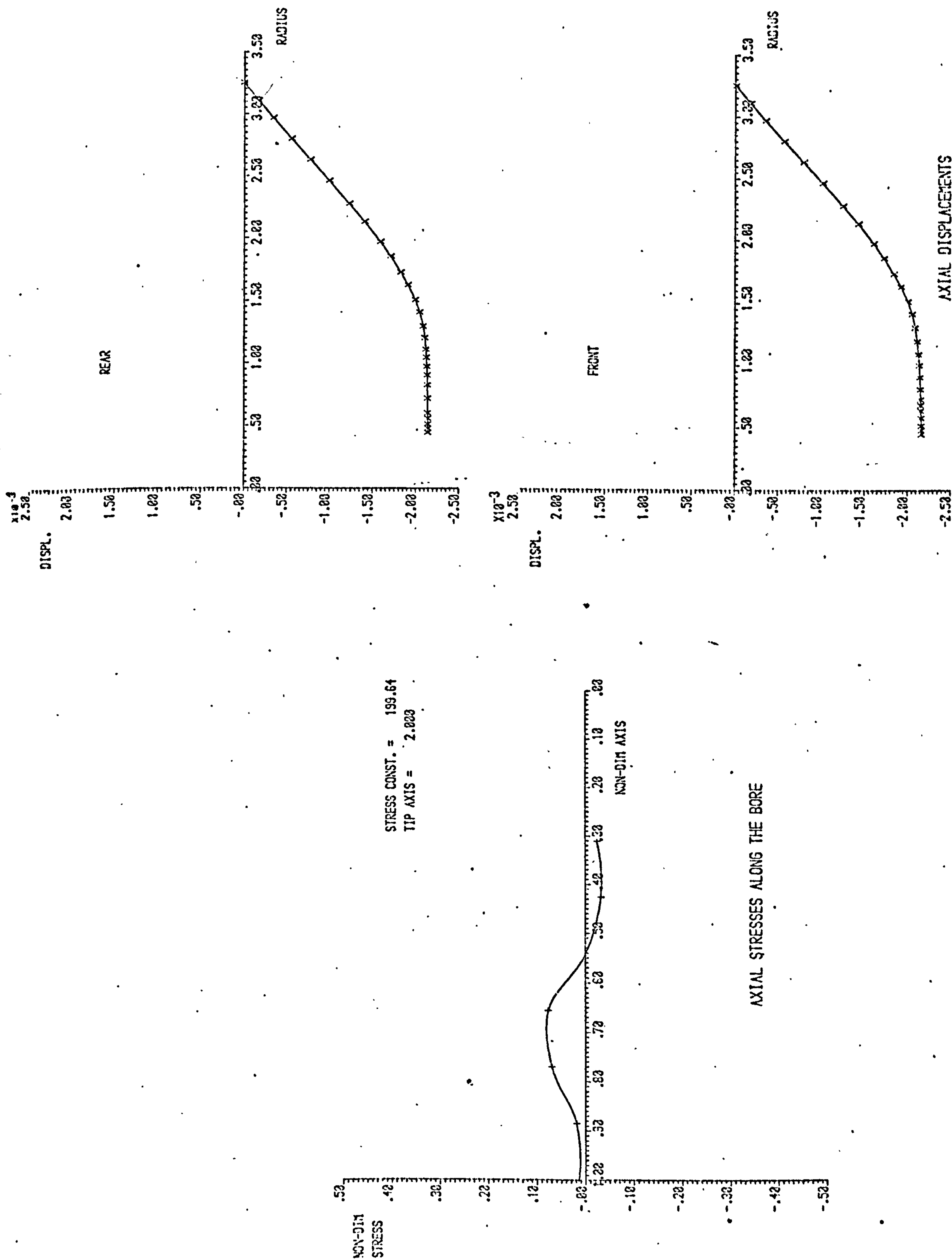
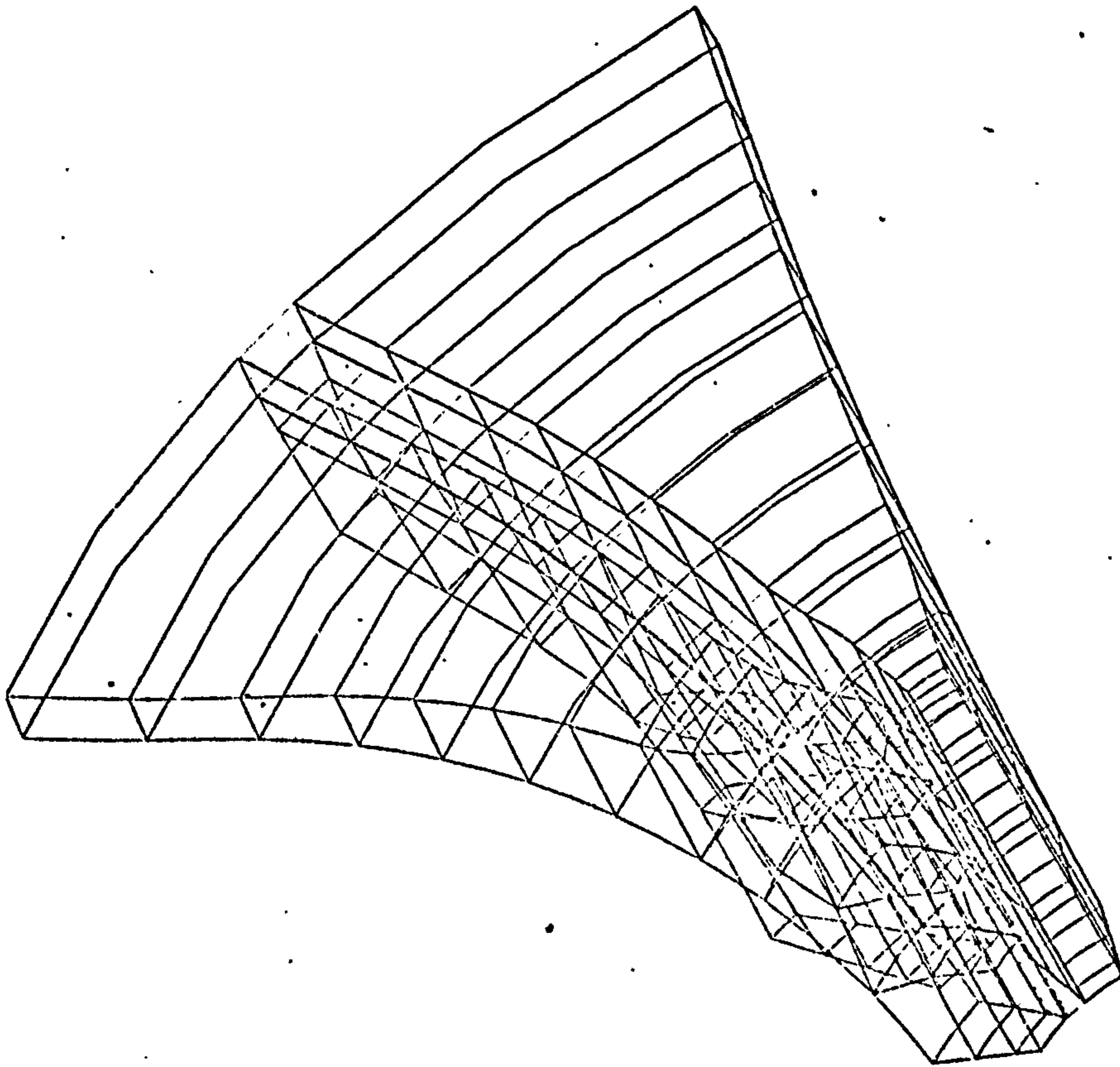


FIG 4-15(a)

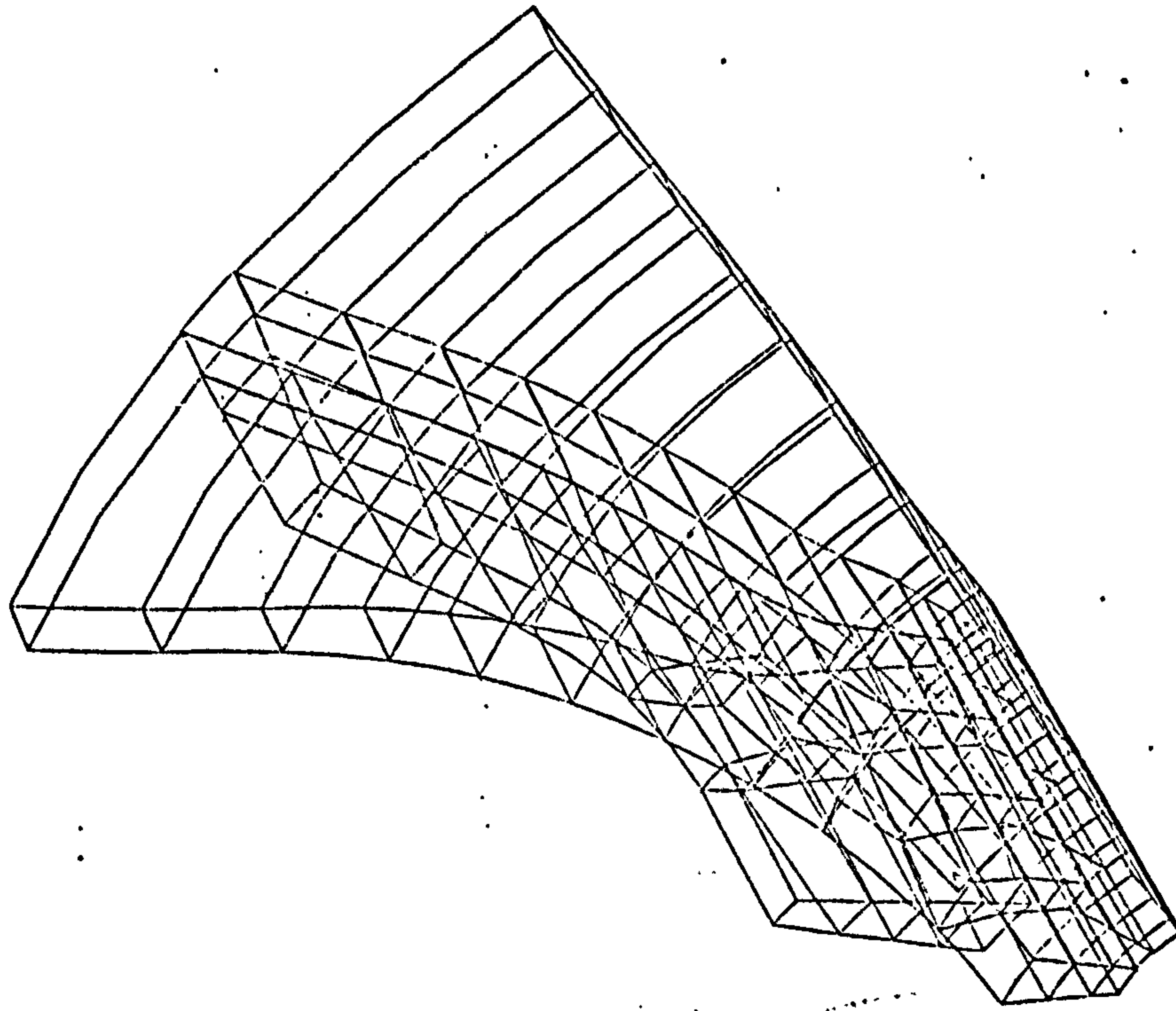






3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN IN SCALE: 55.20 TIMES THE SUPPLIED UNITS

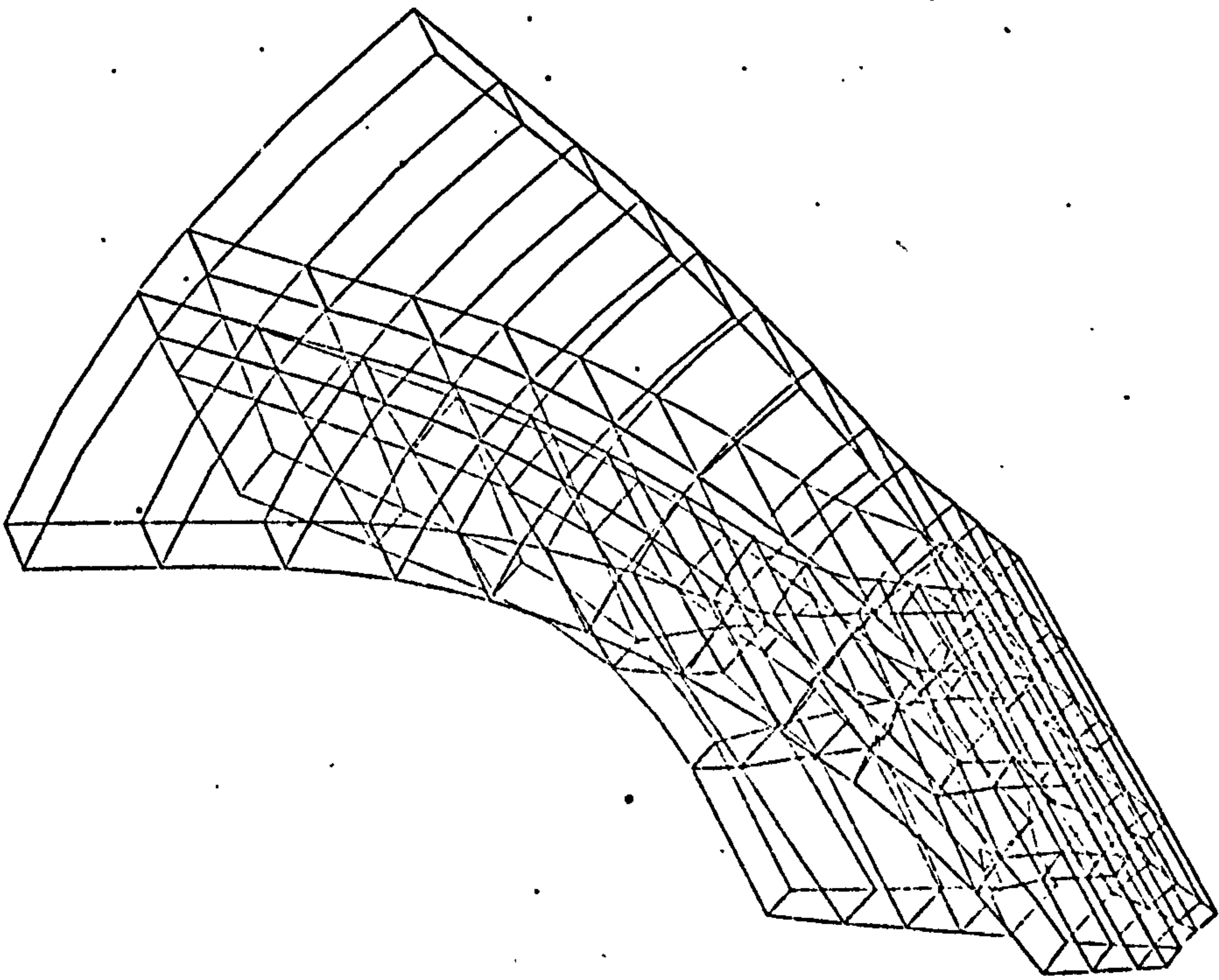
IMPELLER MODEL-P MESH III 7 BLADES



3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN IN SCALE: 55.20 TIMES THE SUPPLIED UNITS

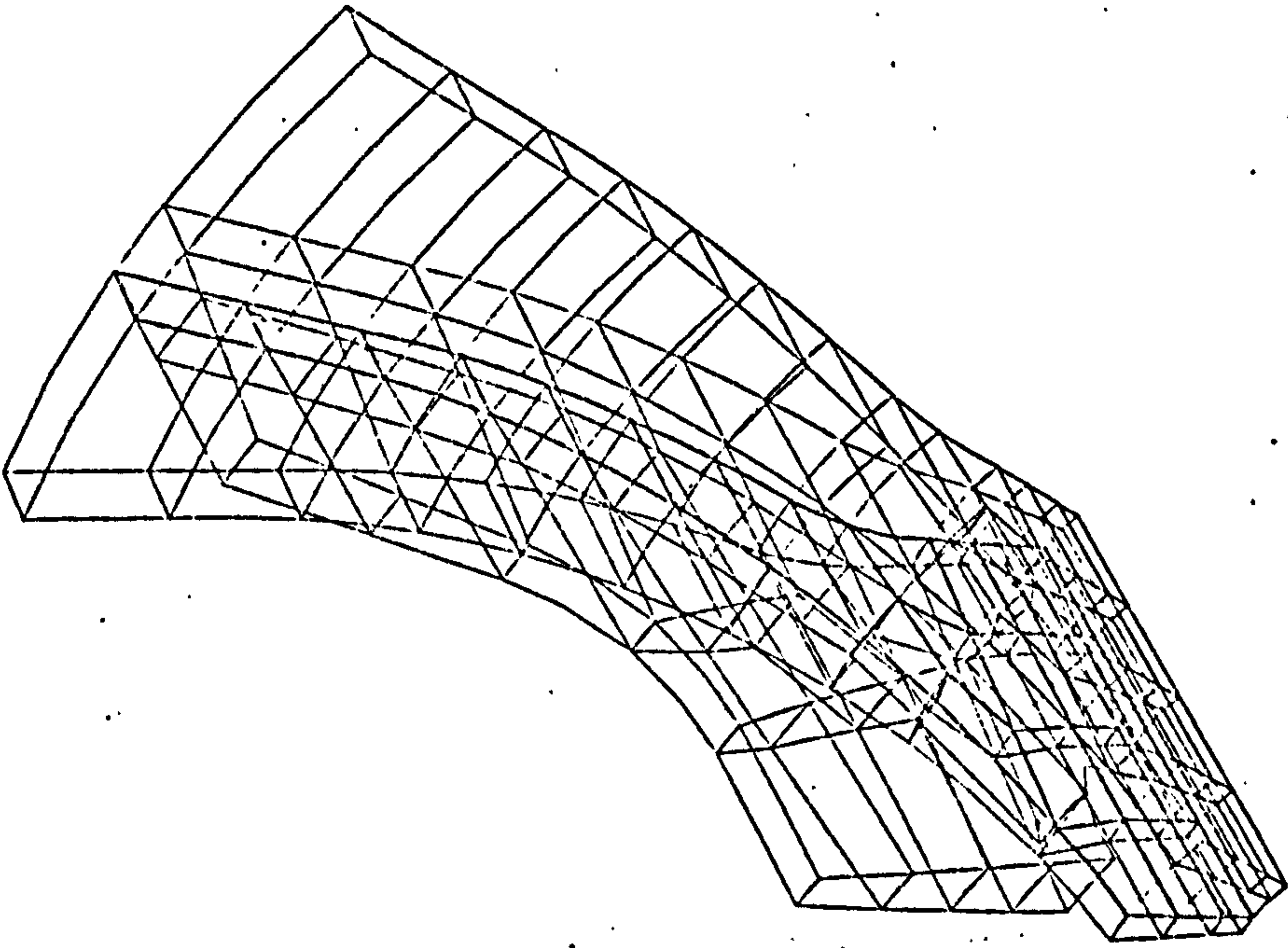
IMPELLER MODEL-P MESH III 9 BLADES

FIG 4-16(a)



3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN INI SCALE: 55.00 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-P MESH III 11 BLADES



3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN INI SCALE: 55.00 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-P MESH III 13 BLADES

FIG 4-16(b)

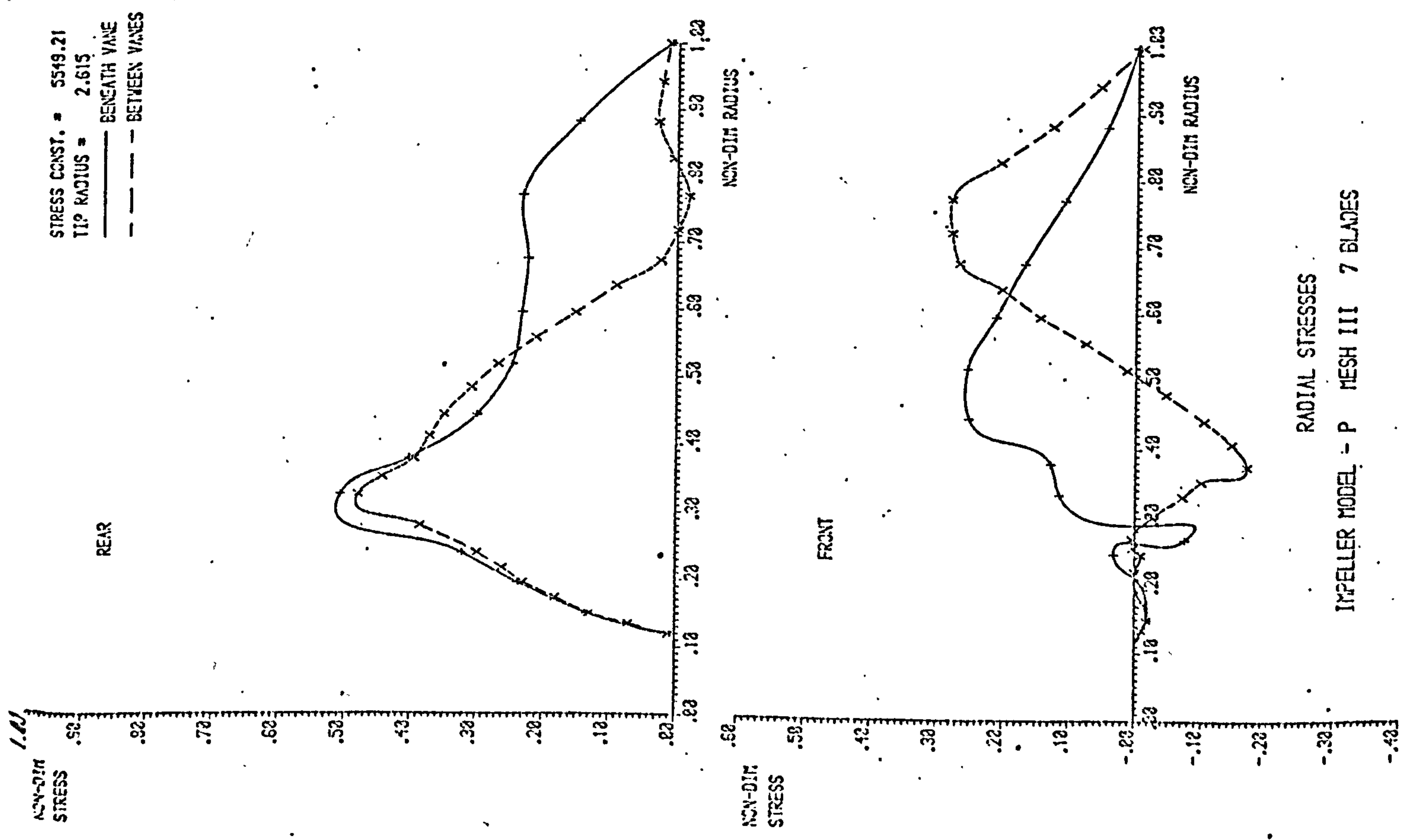
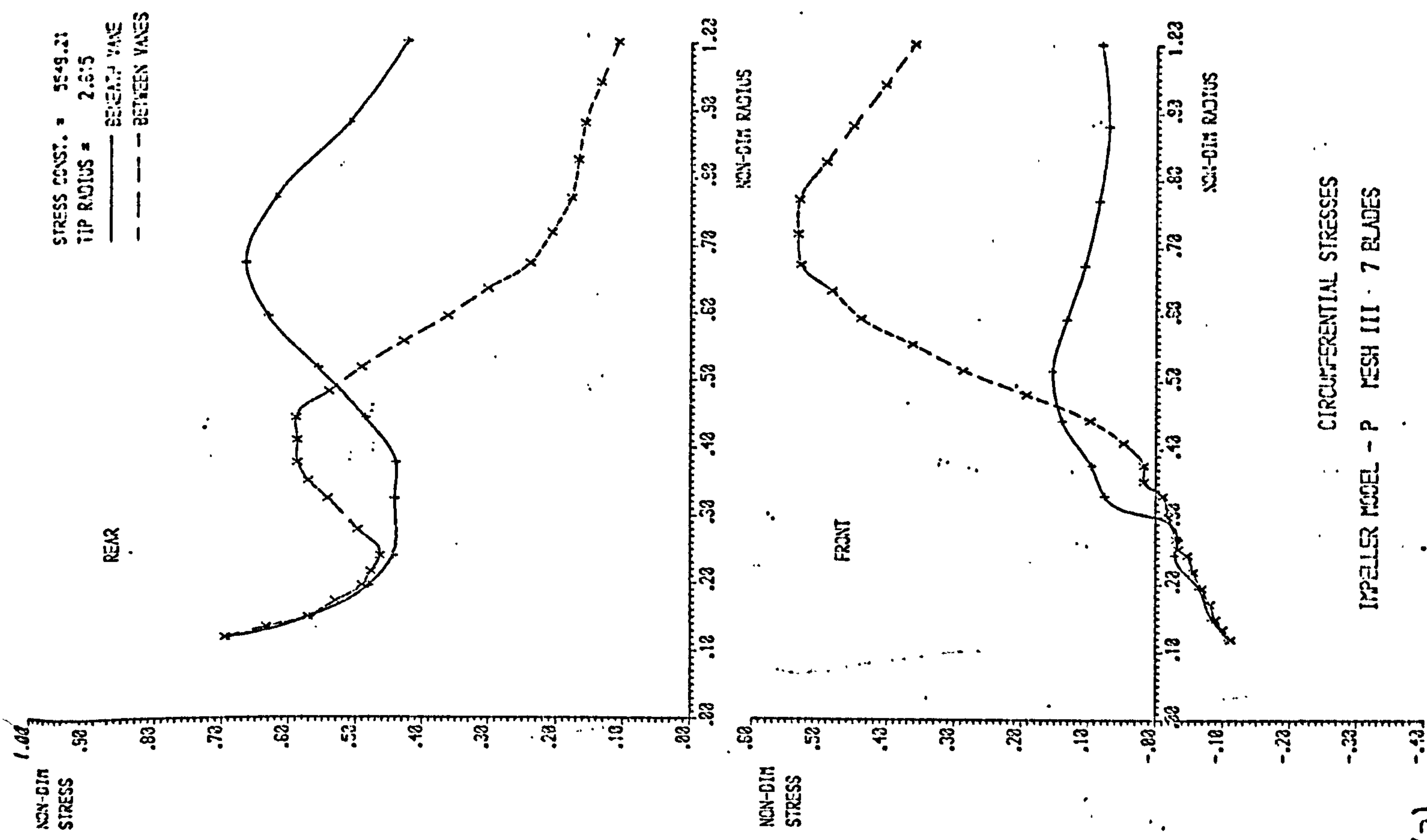
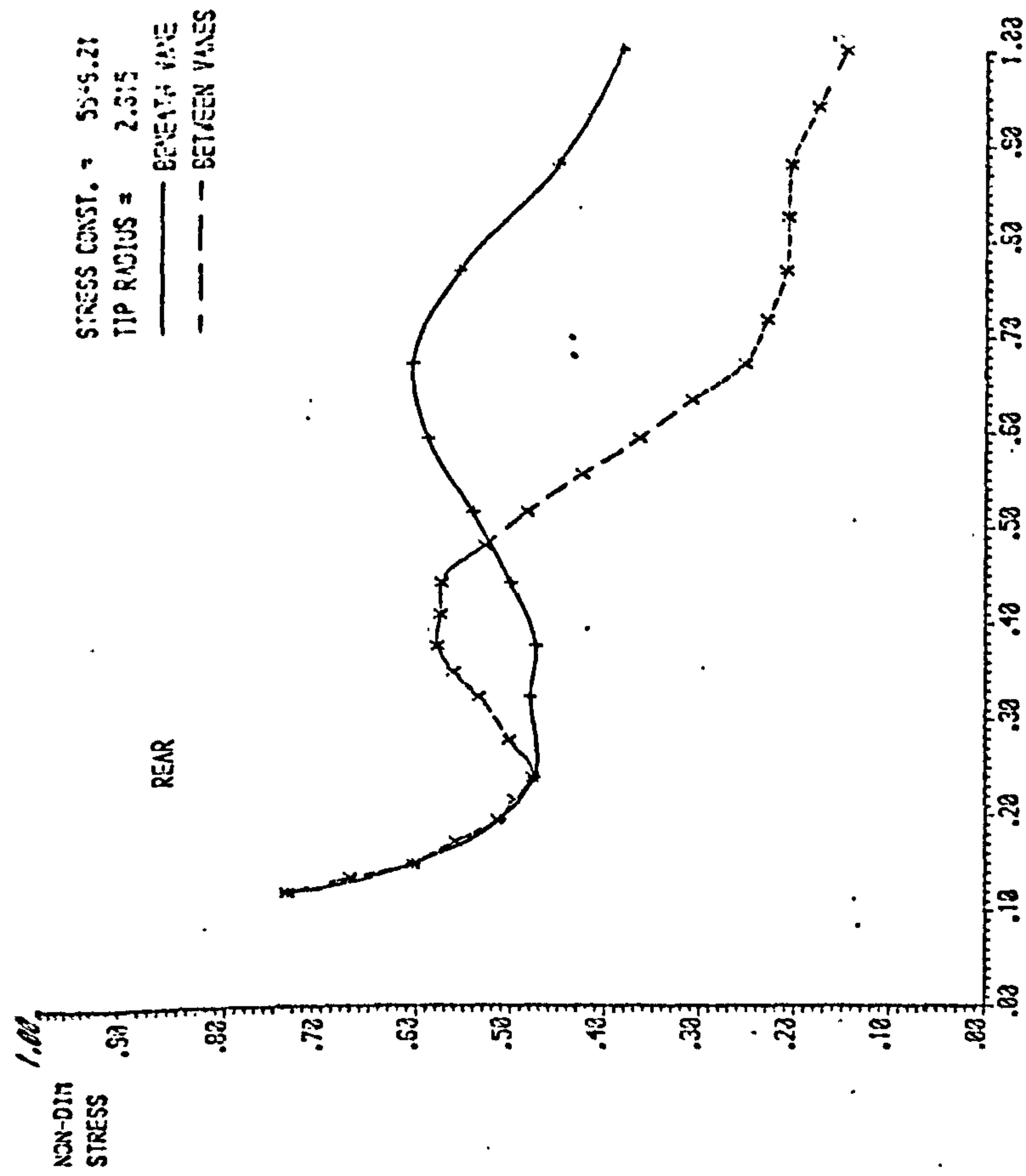
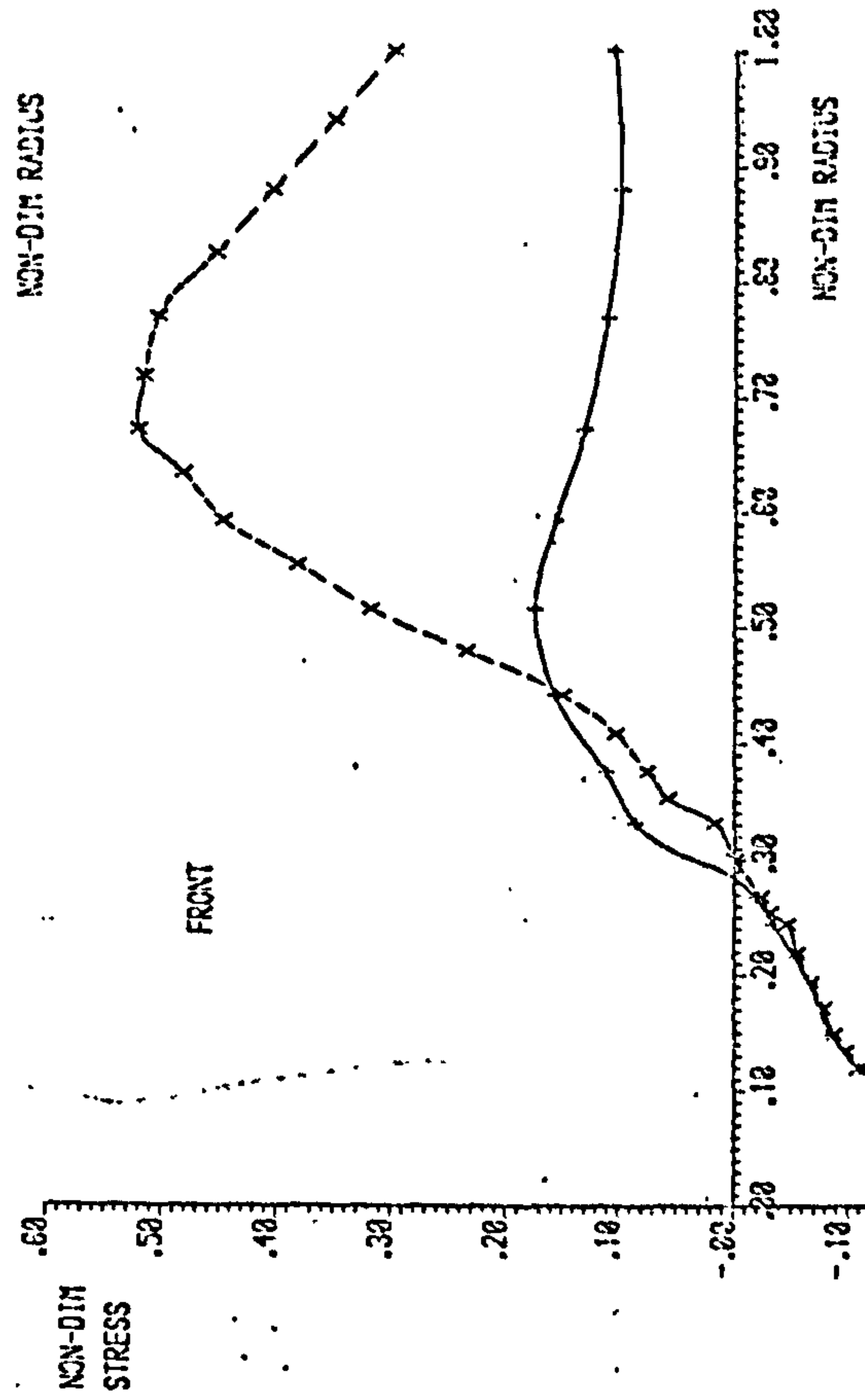
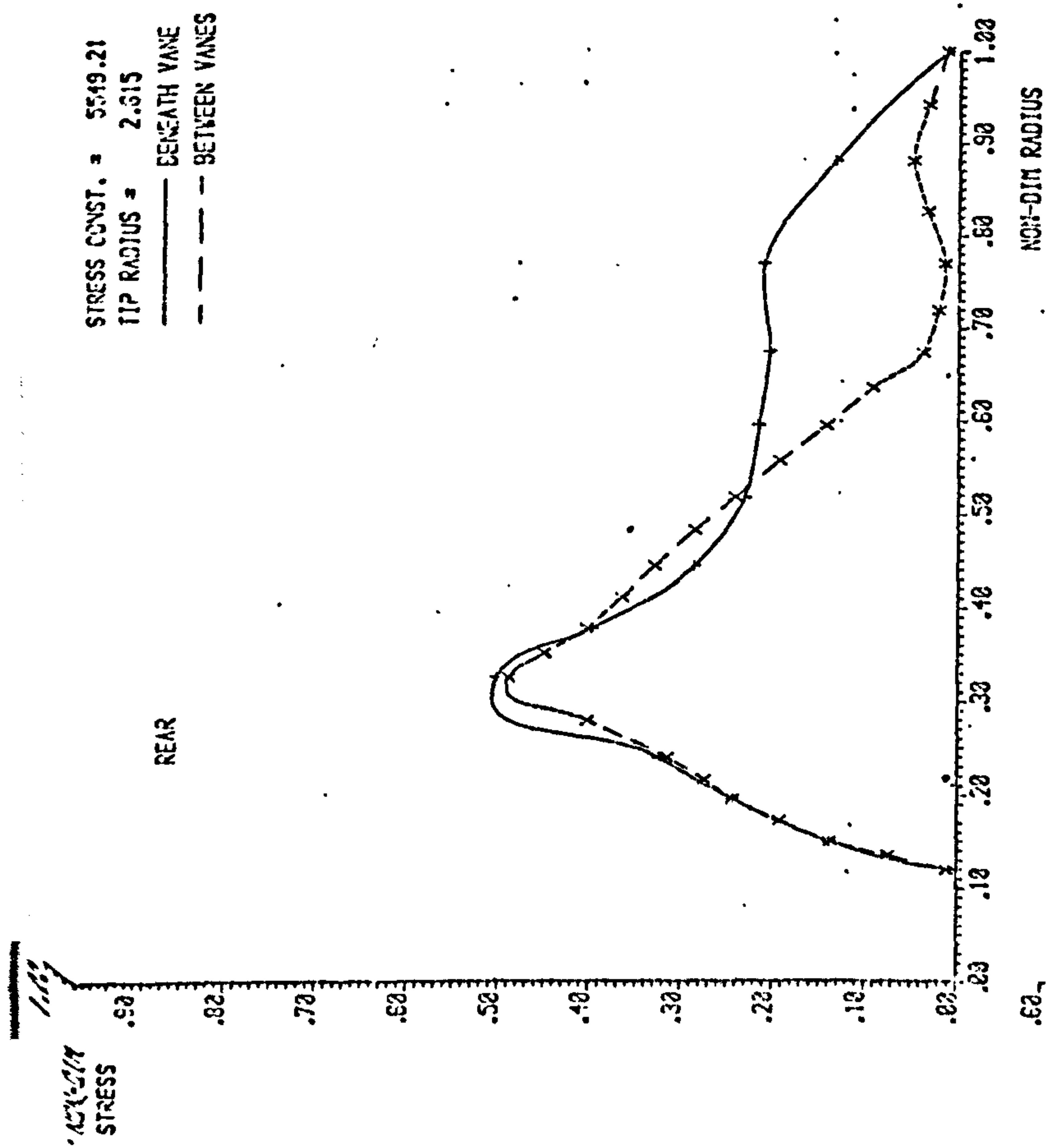
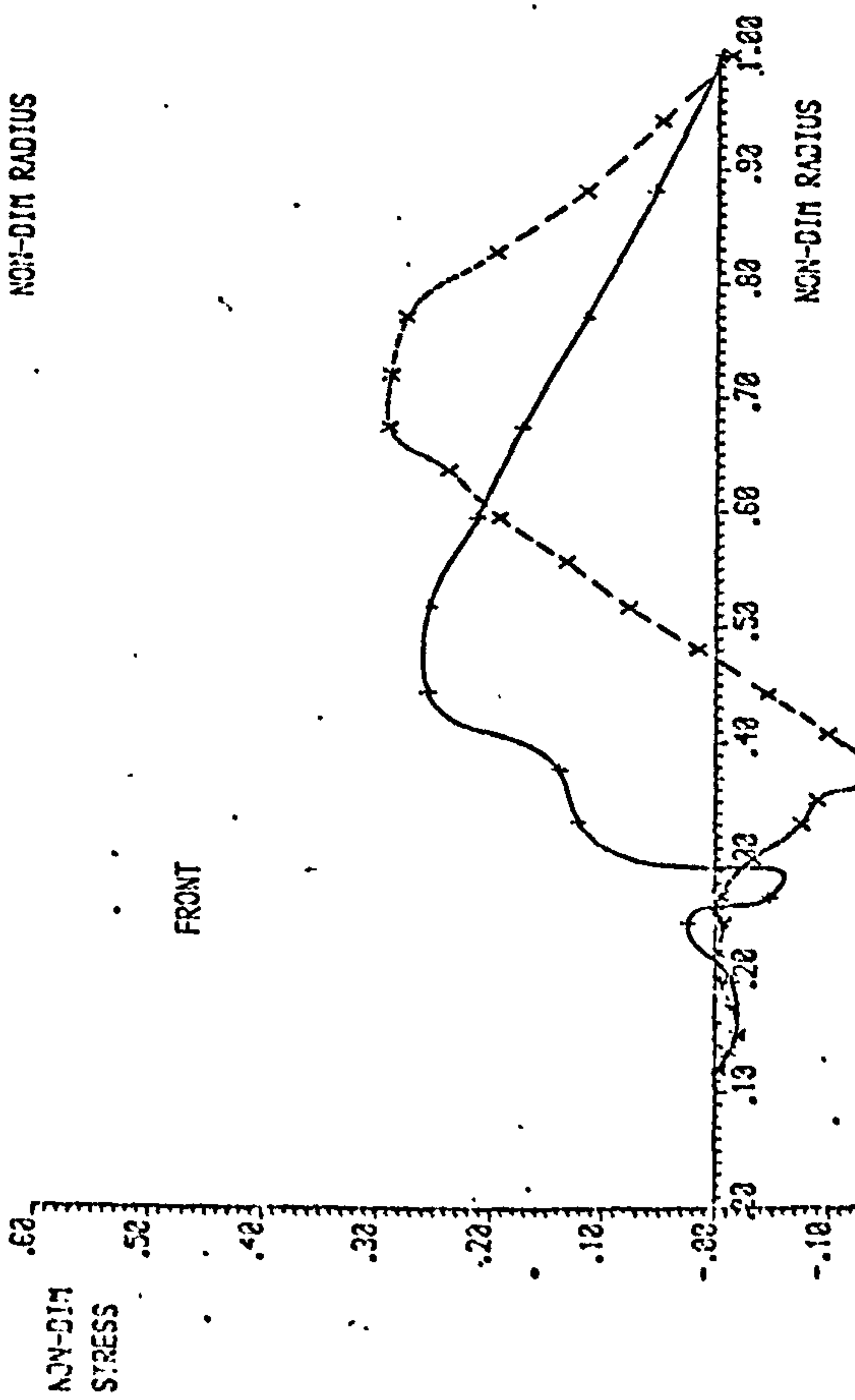


FIG 4.17(a)

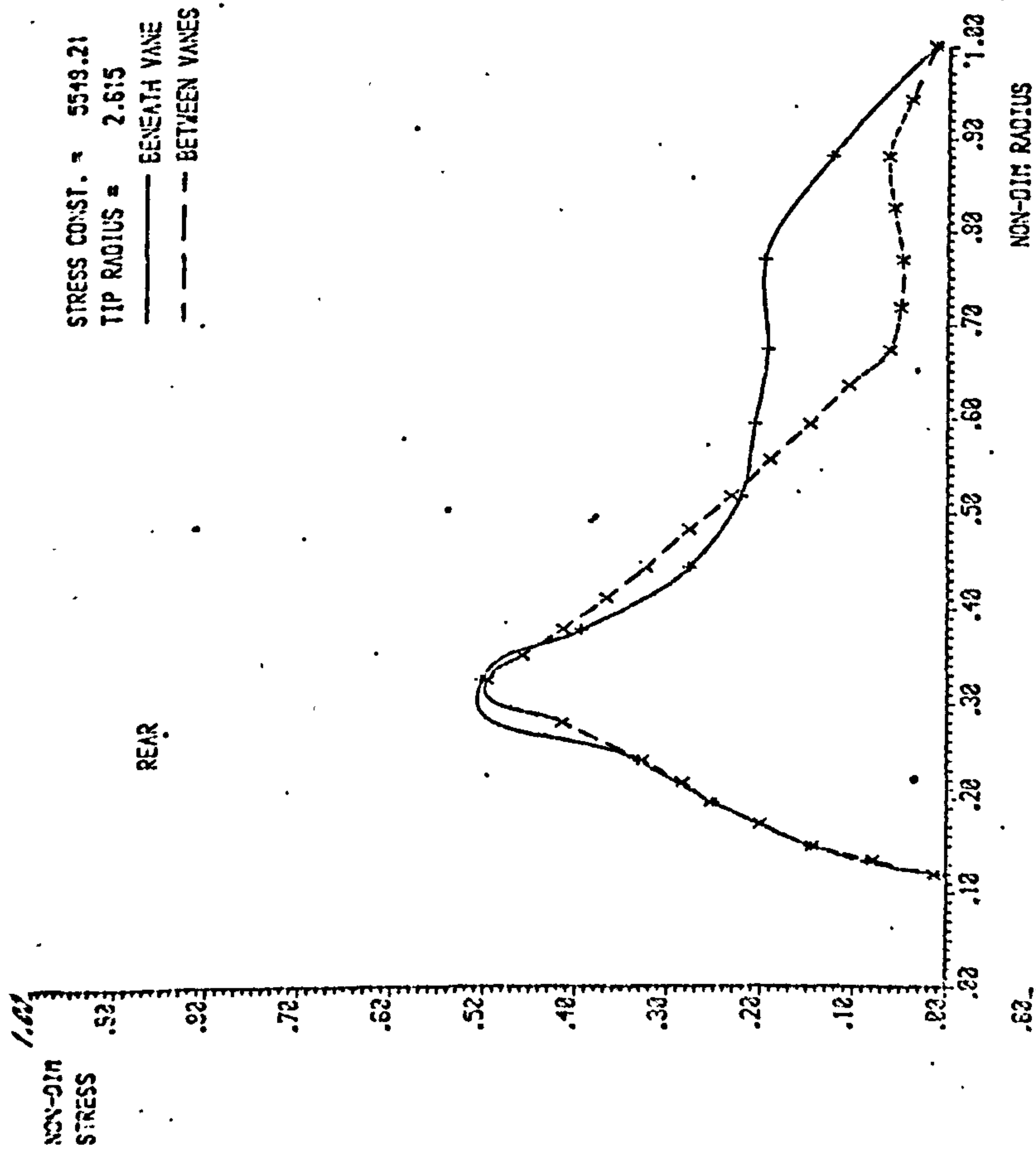
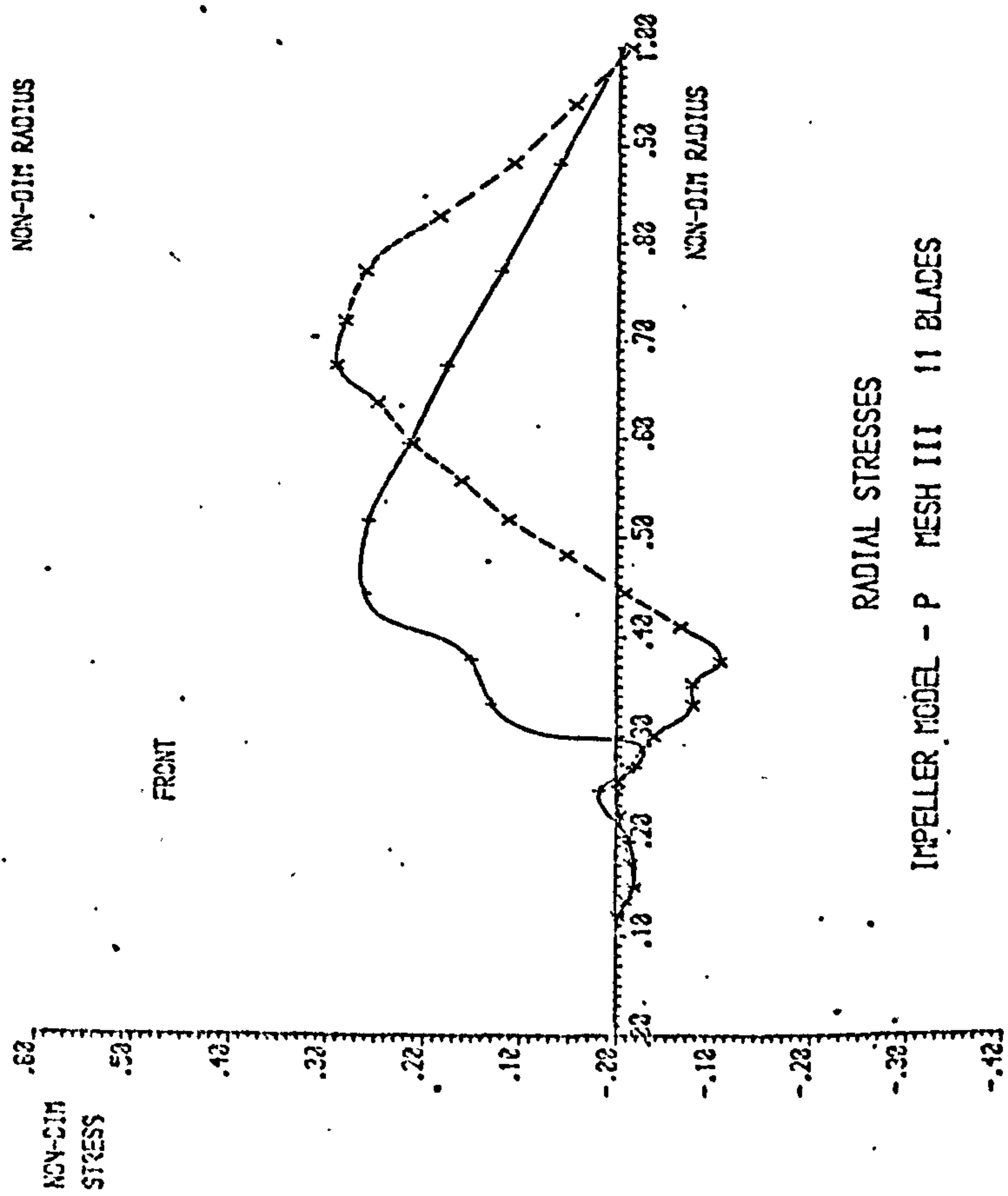


CIRCUMFERENTIAL STRESSES
IMPELLER MODEL - P MESH III 9 BLADES

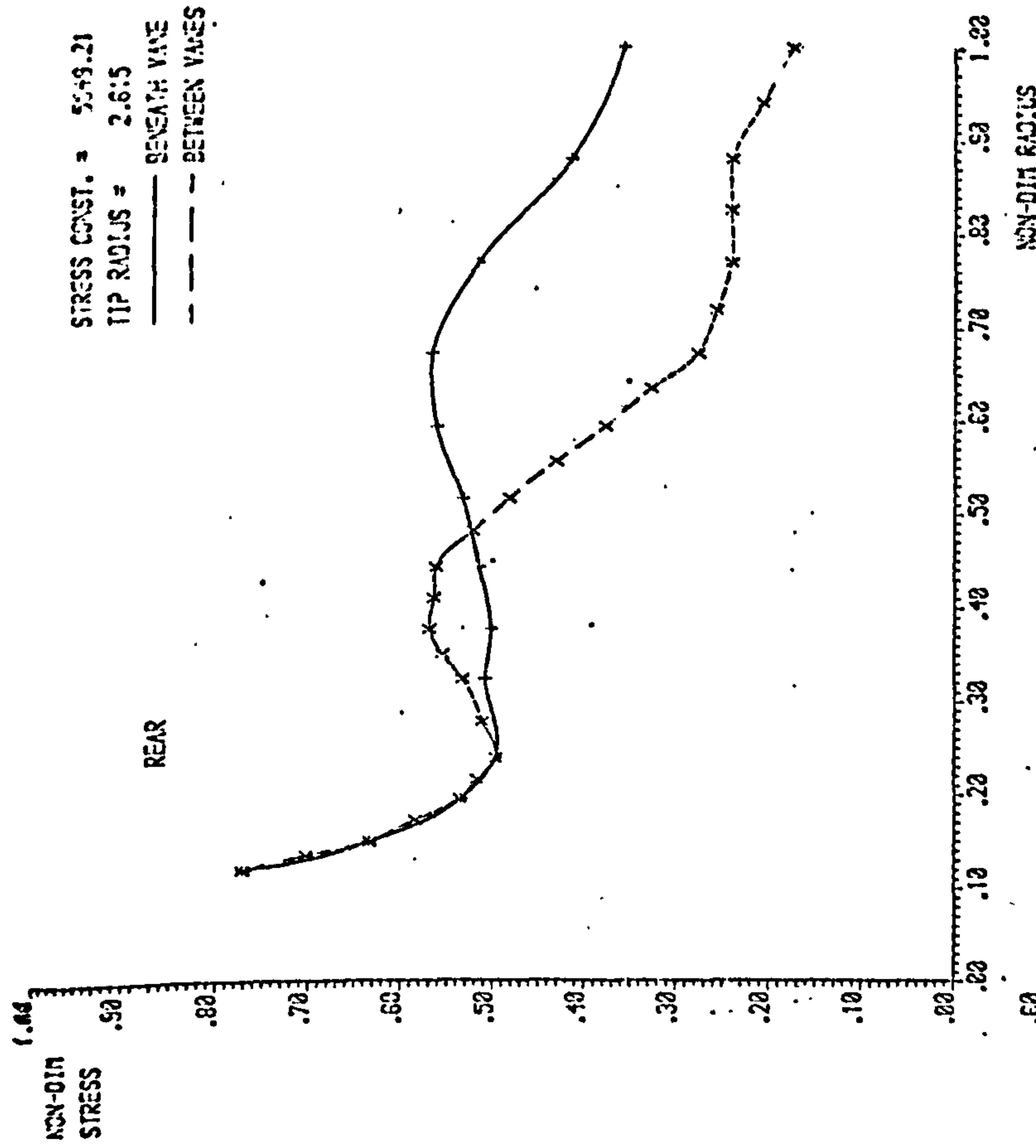
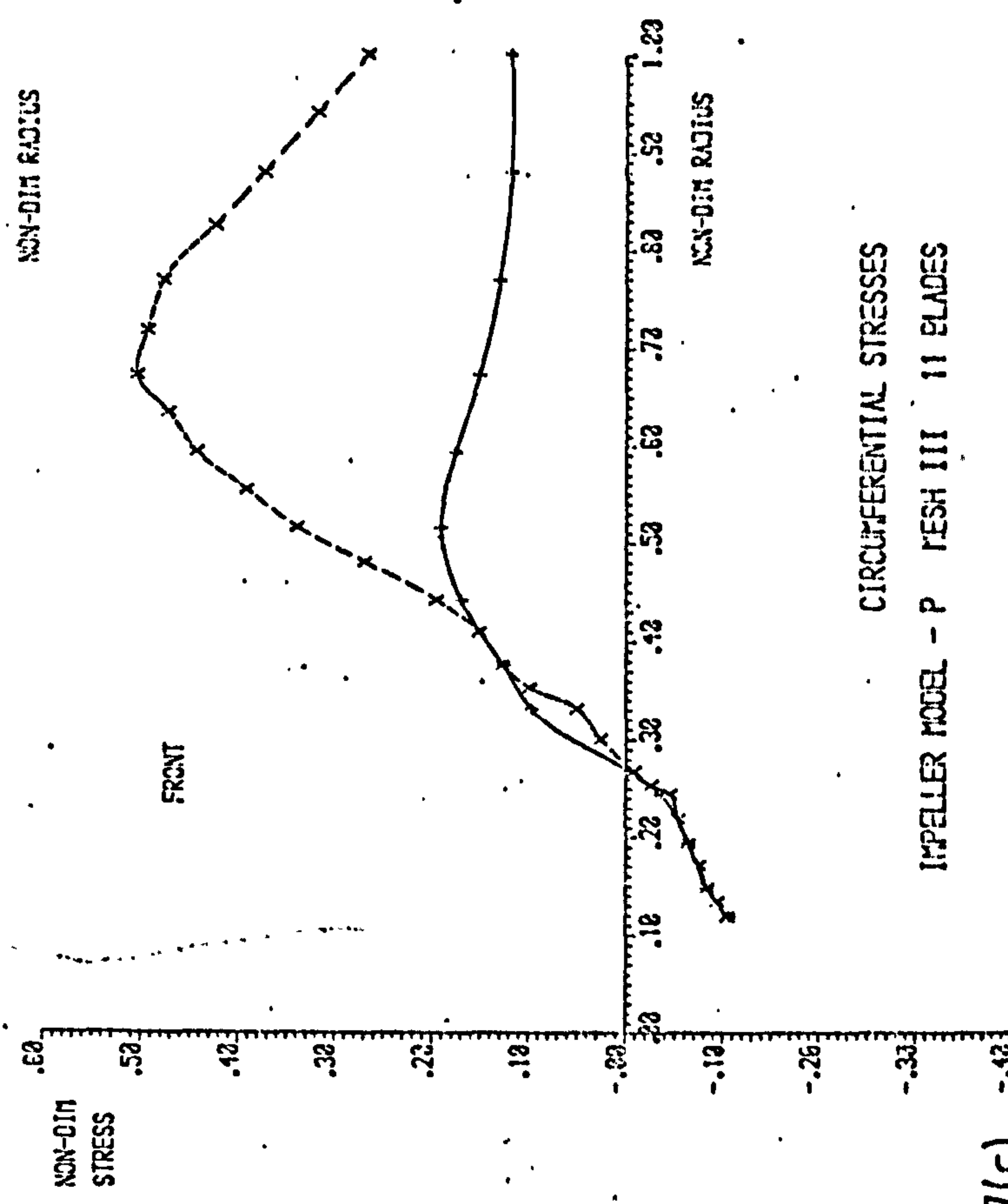


RADIAL STRESSES
IMPELLER MODEL - P MESH III 9 BLADES

FIG 4-17(b)

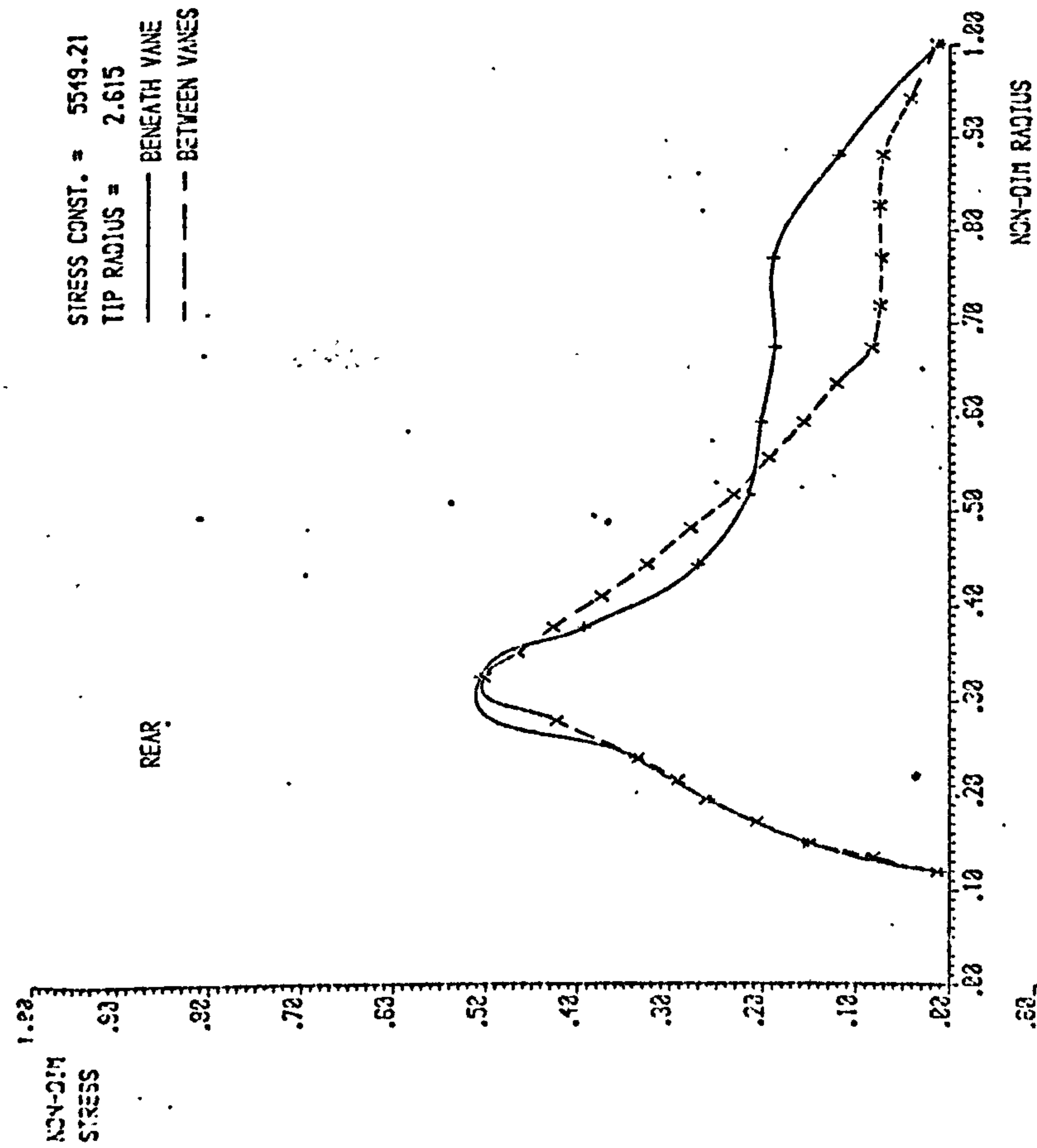
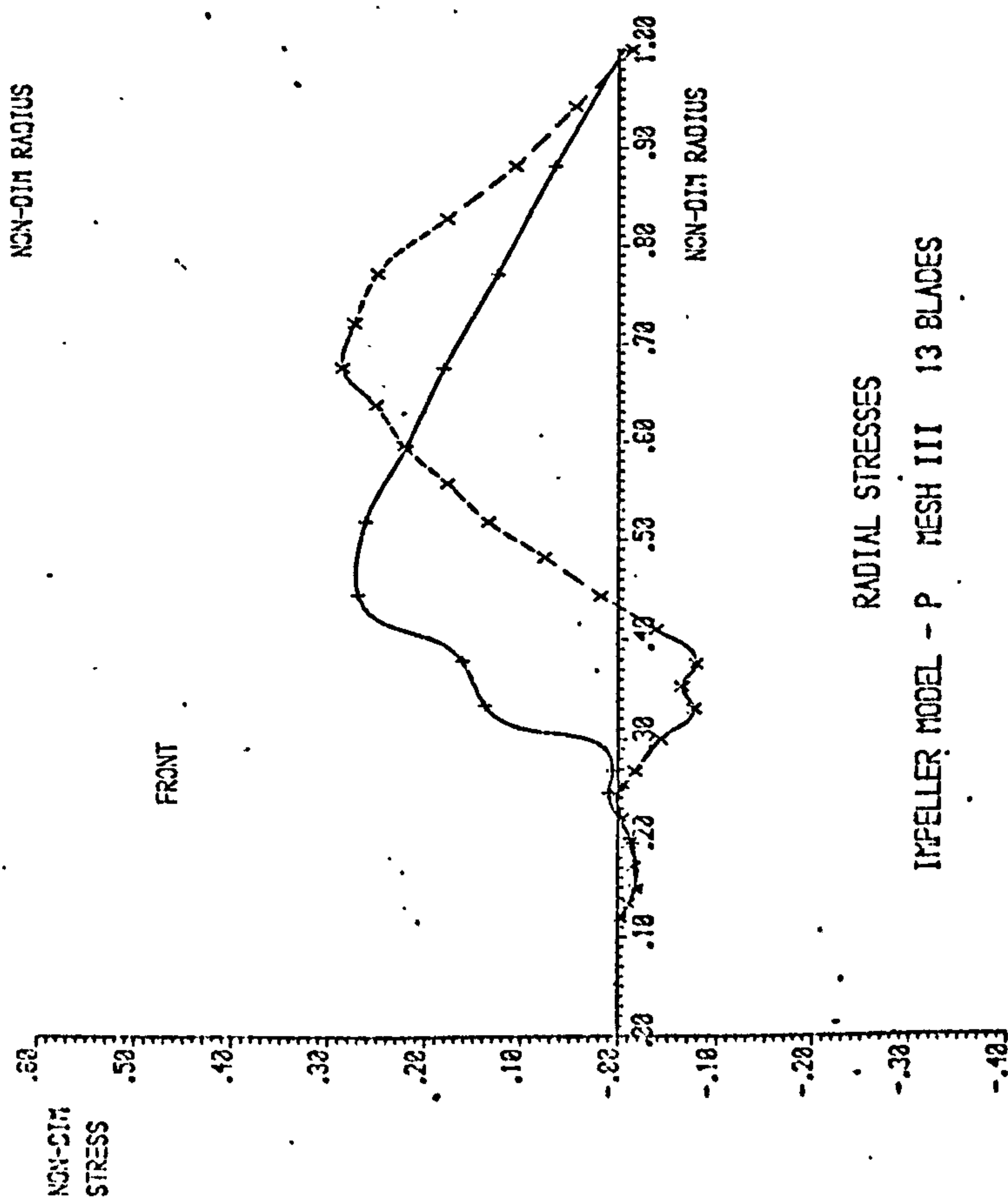


RADIAL STRESSES
IMPELLER MODEL - P MESH III 11 BLADES

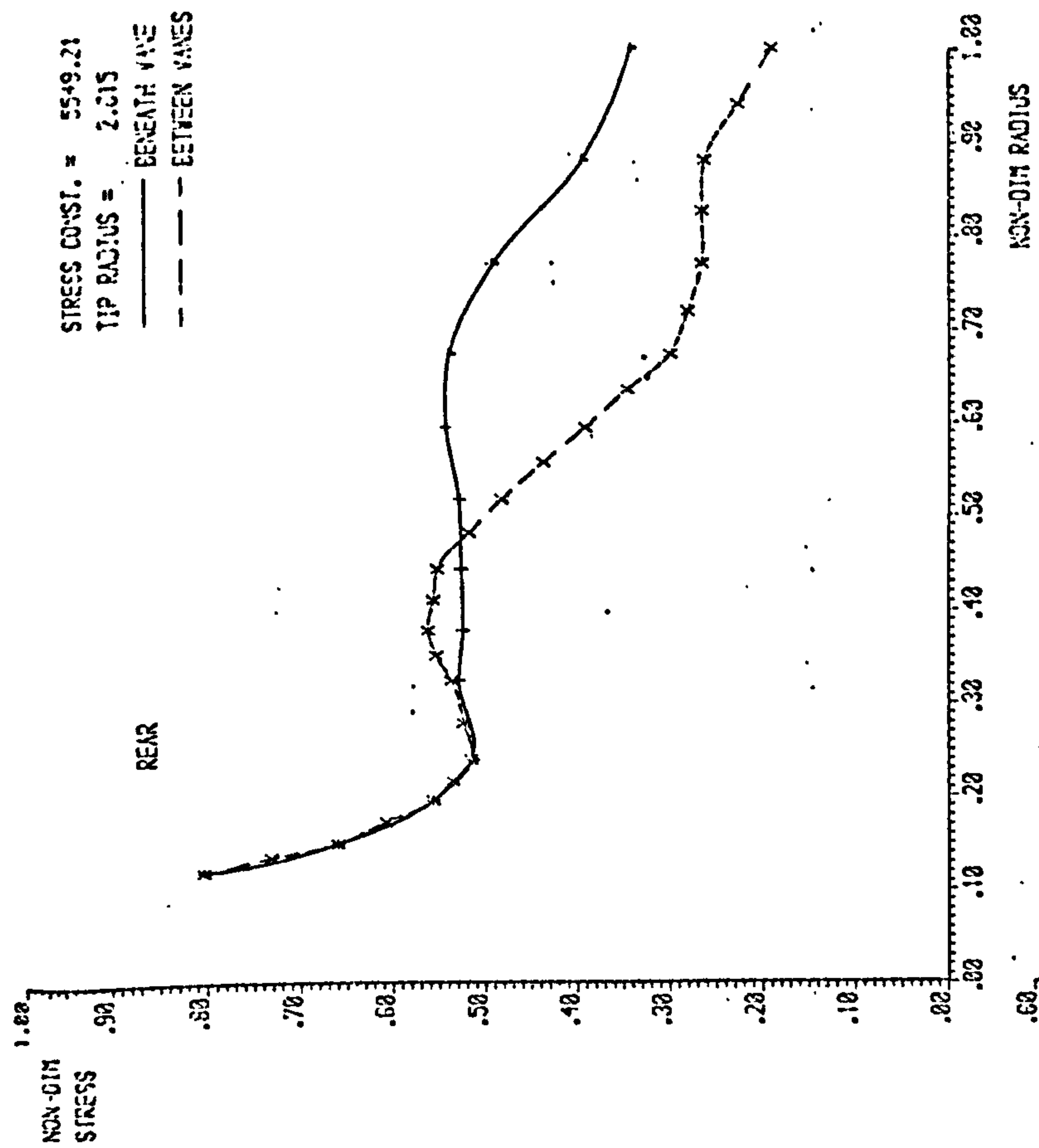
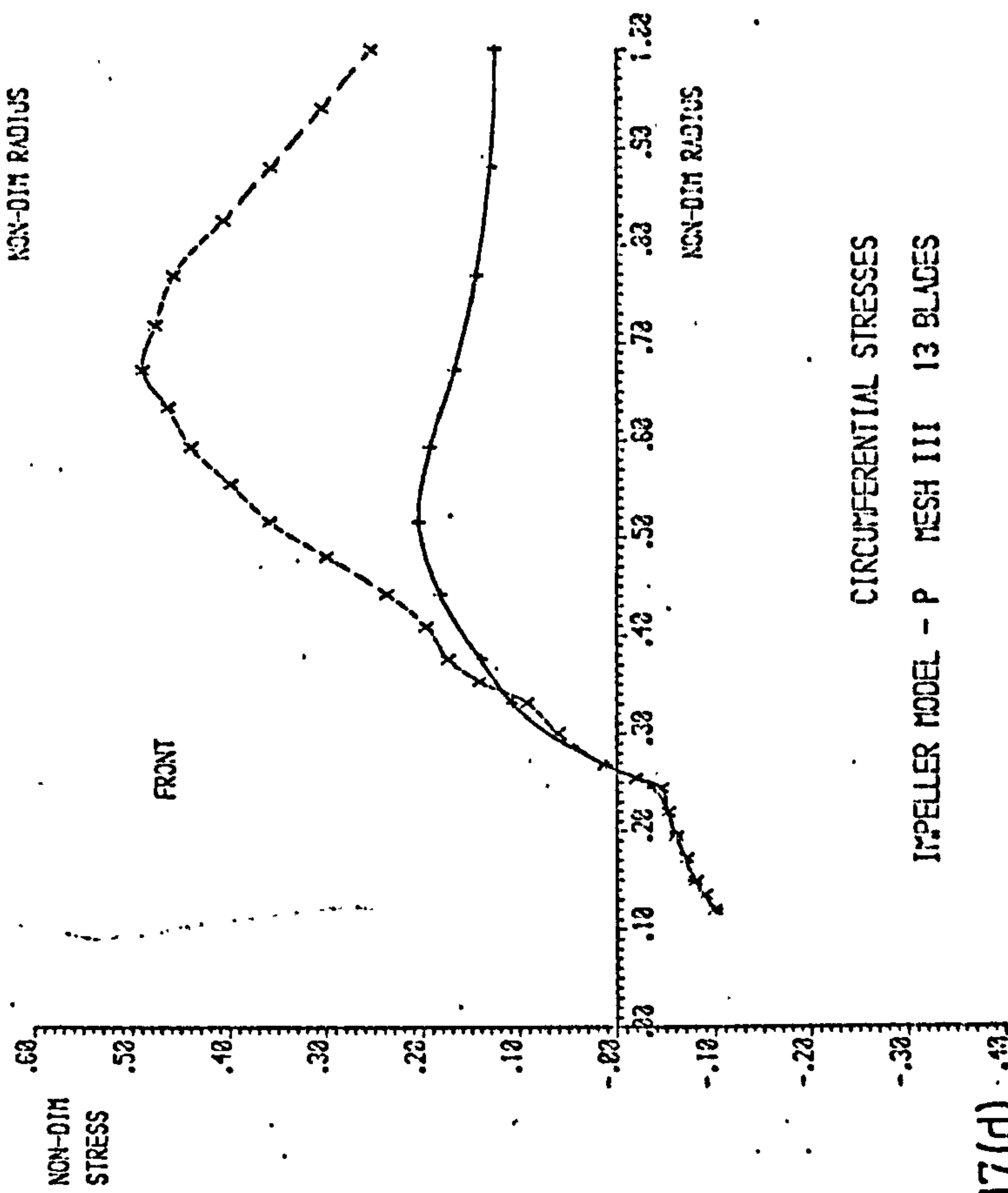


CIRCUMFERENTIAL STRESSES
IMPELLER MODEL - P MESH III 11 BLADES

FIG 4-17(c)

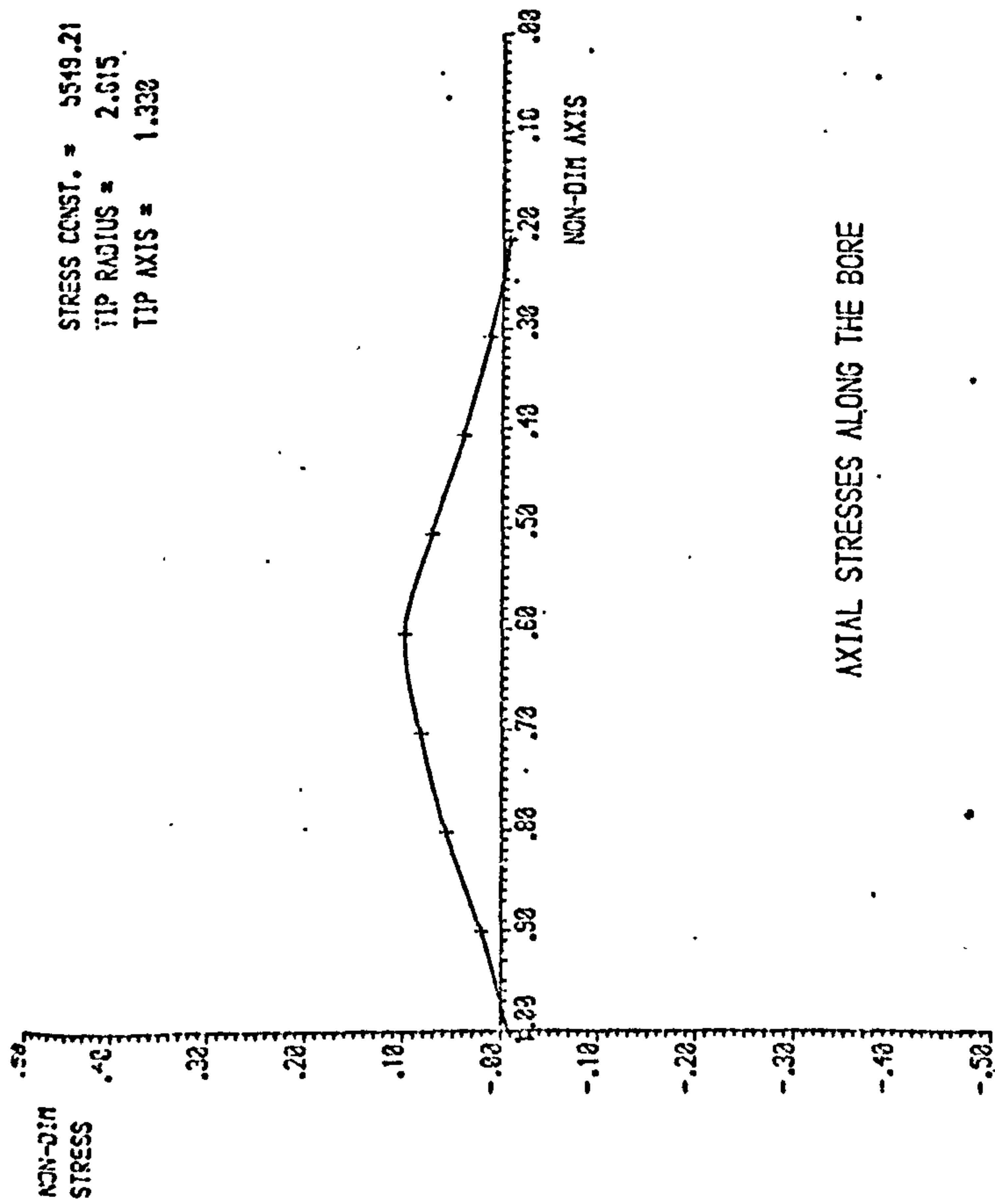


RADIAL STRESSES
IMPELLER MODEL - P MESH III 13 BLADES

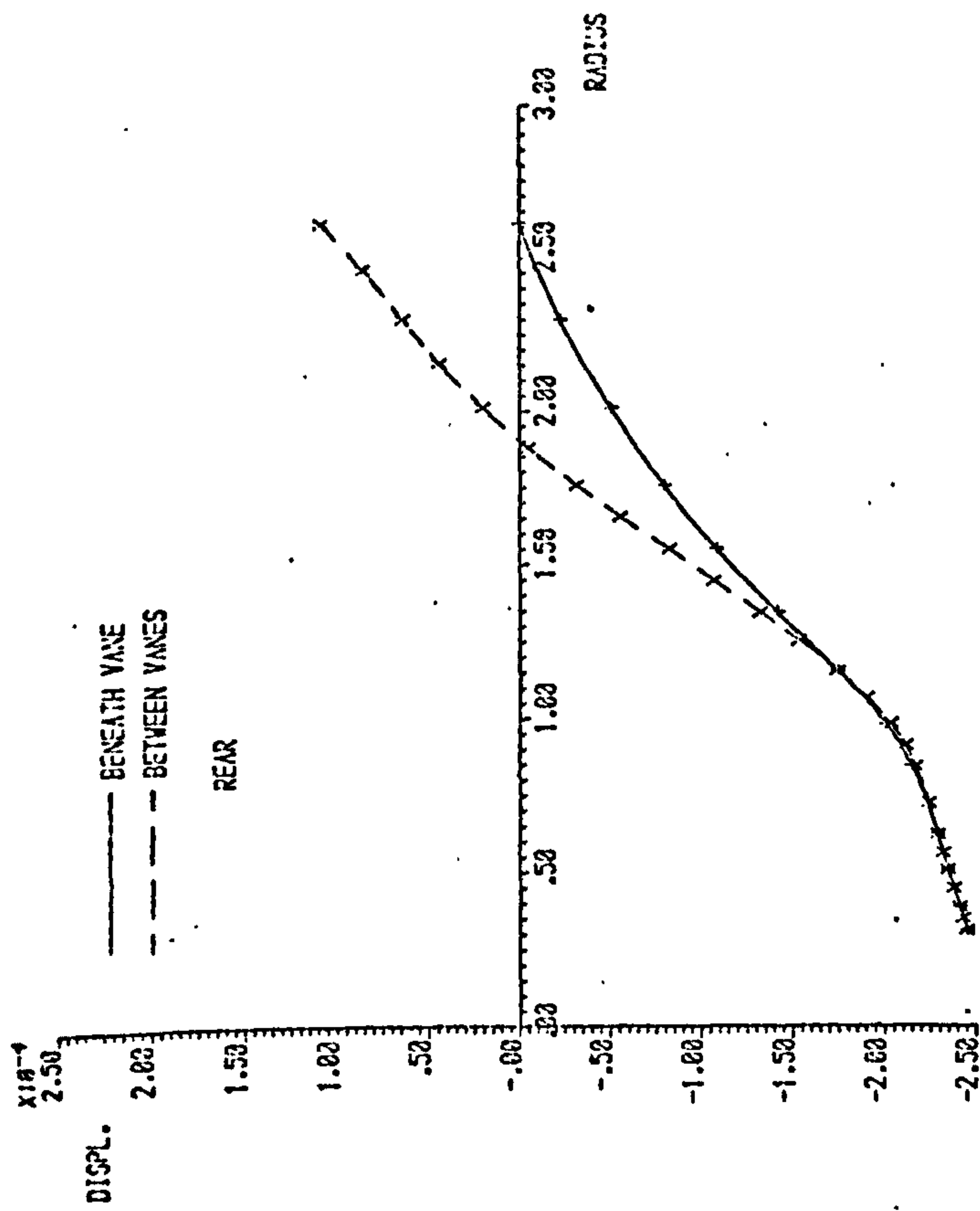
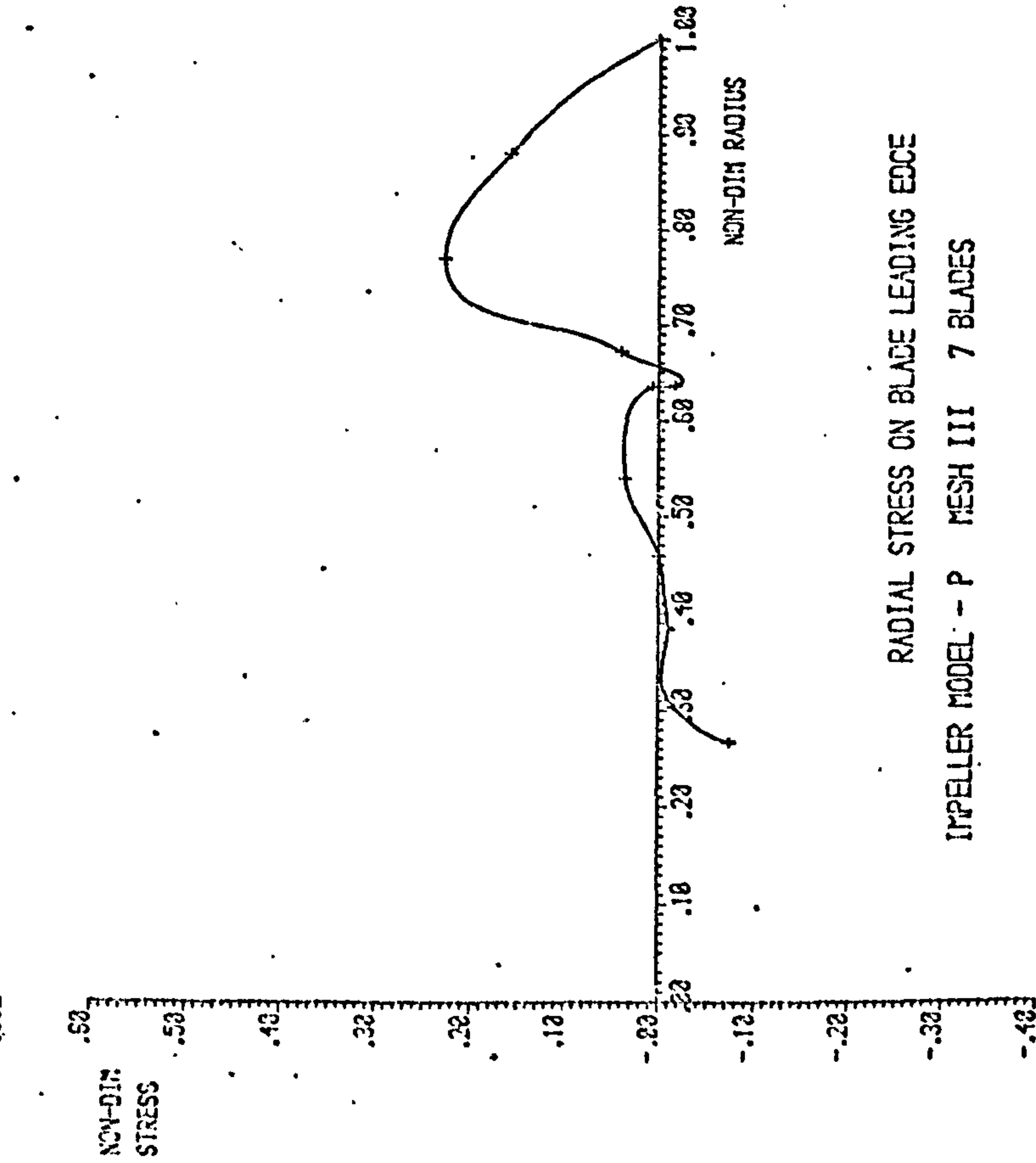


CIRCUMFERENTIAL STRESSES
IMPELLER MODEL - P MESH III 13 BLADES

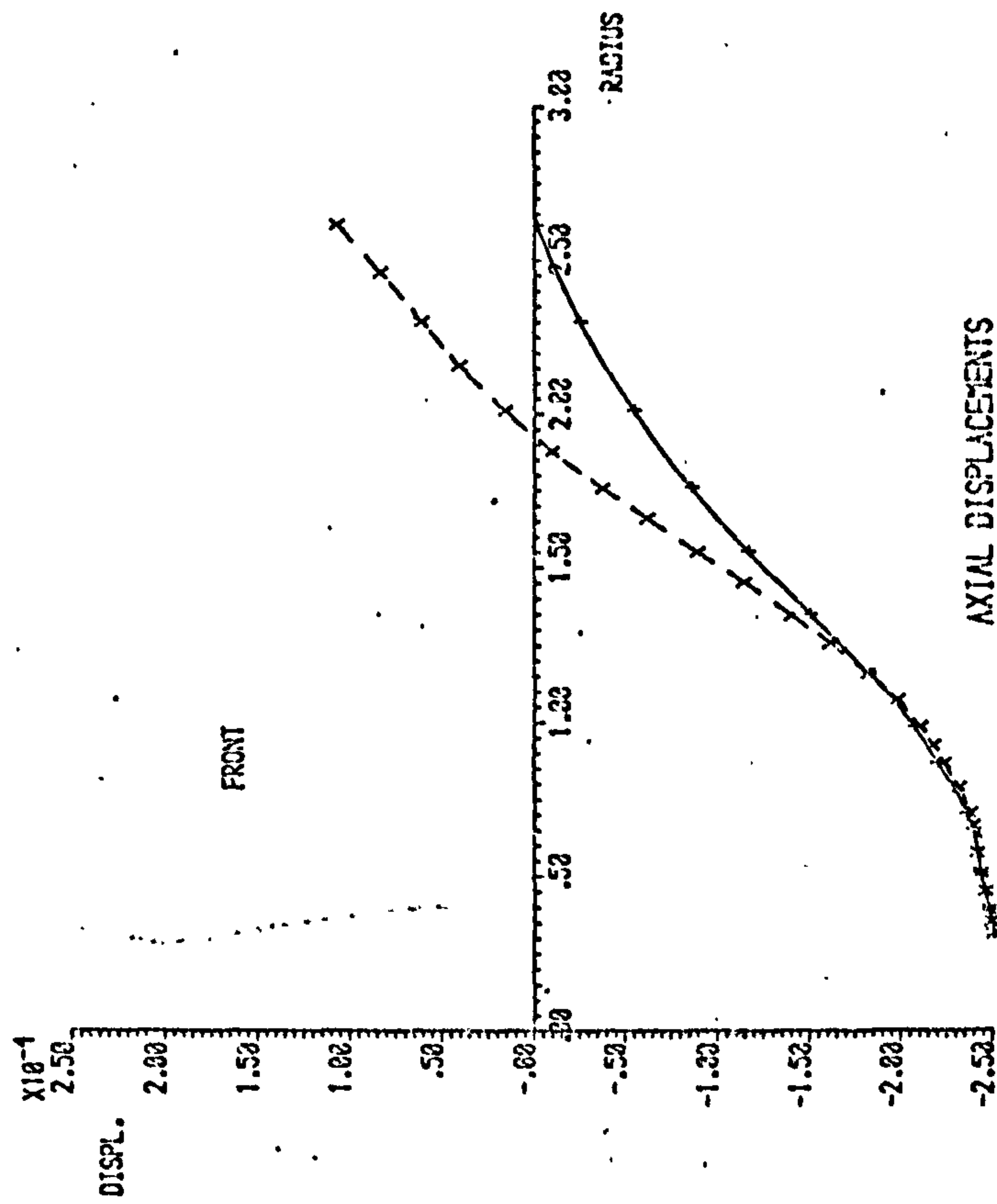
FIG 4-17(d)



AXIAL STRESSES ALONG THE BORE



RADIAL STRESS ON BLADE LEADING EDGE

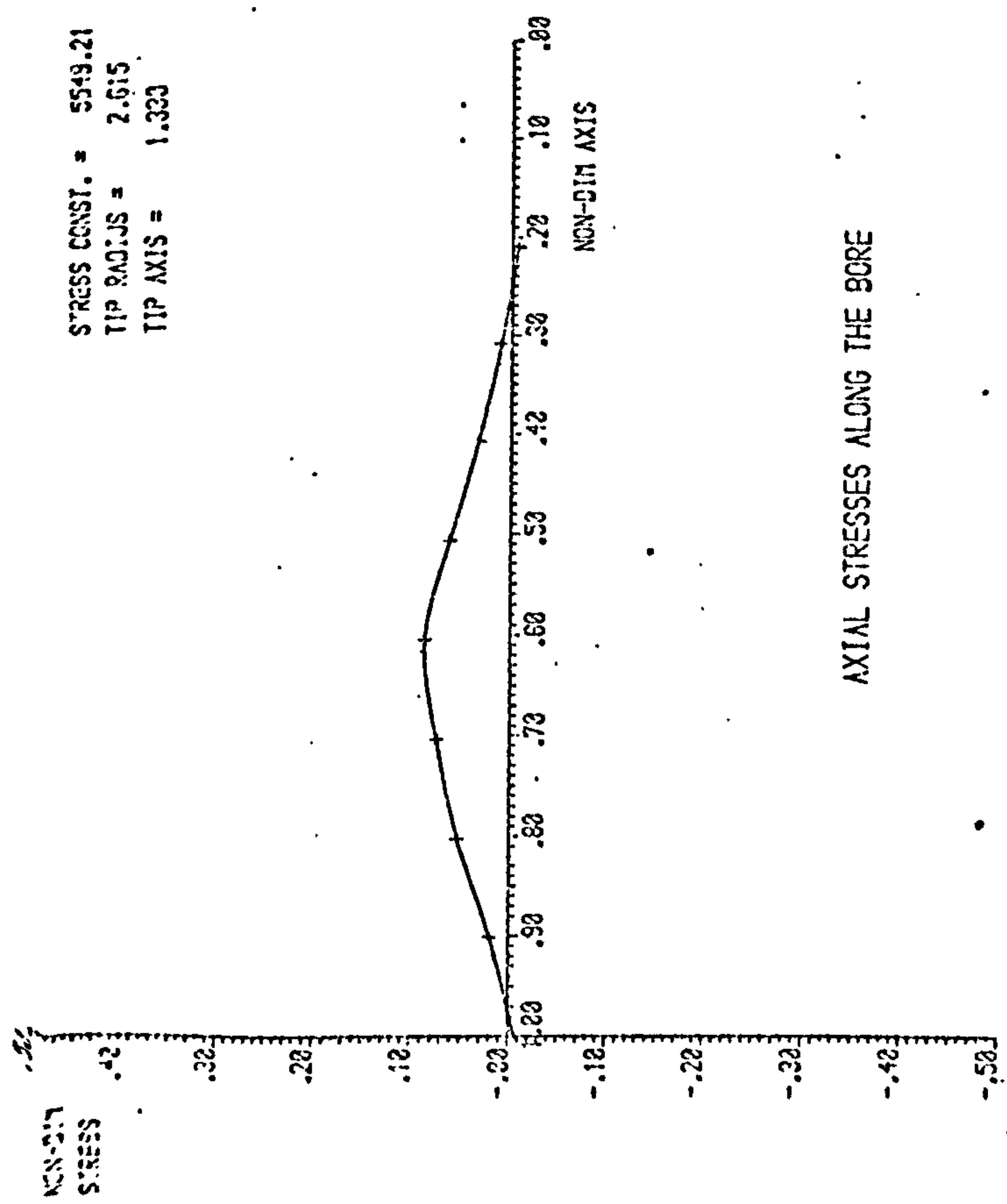
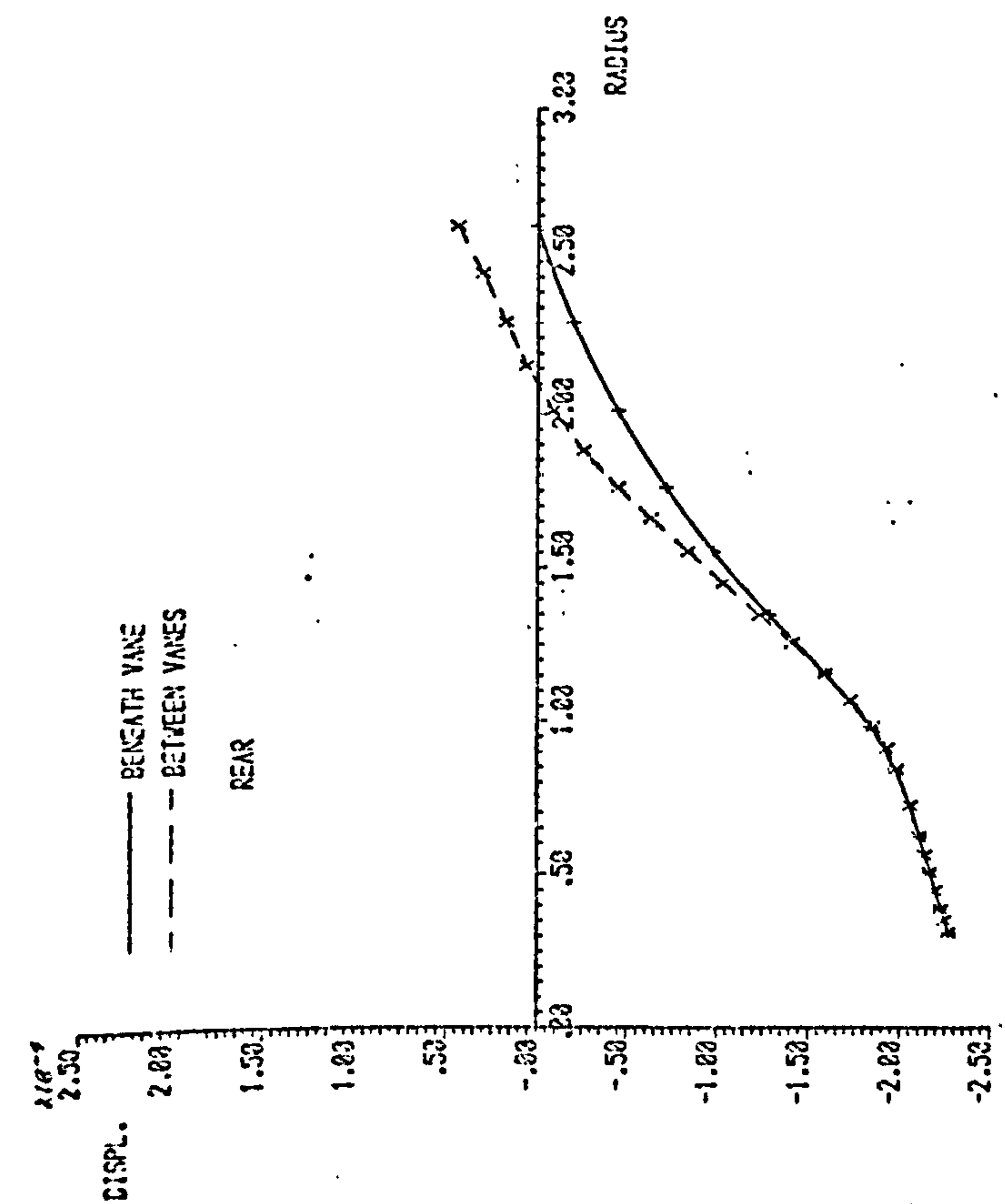
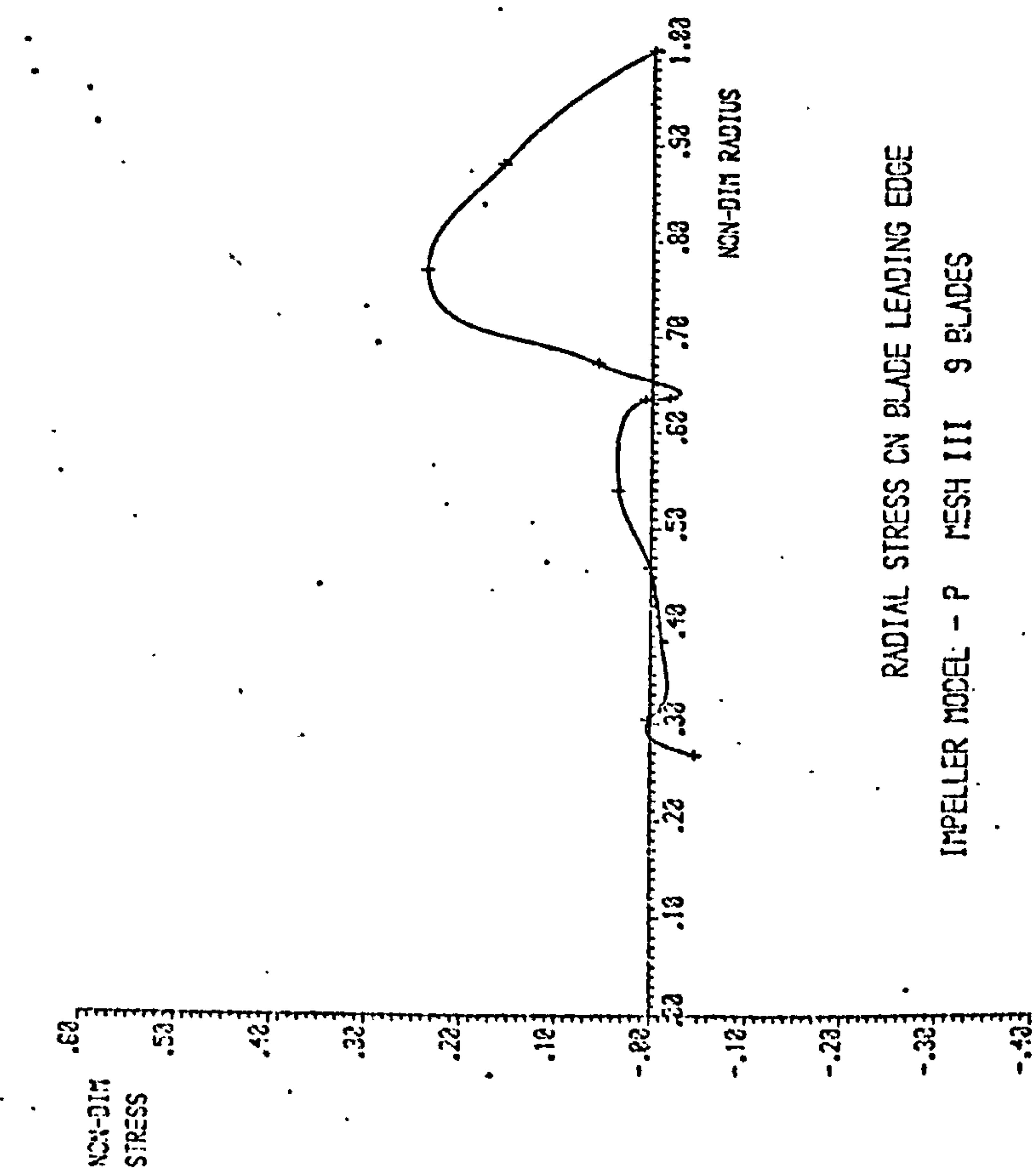
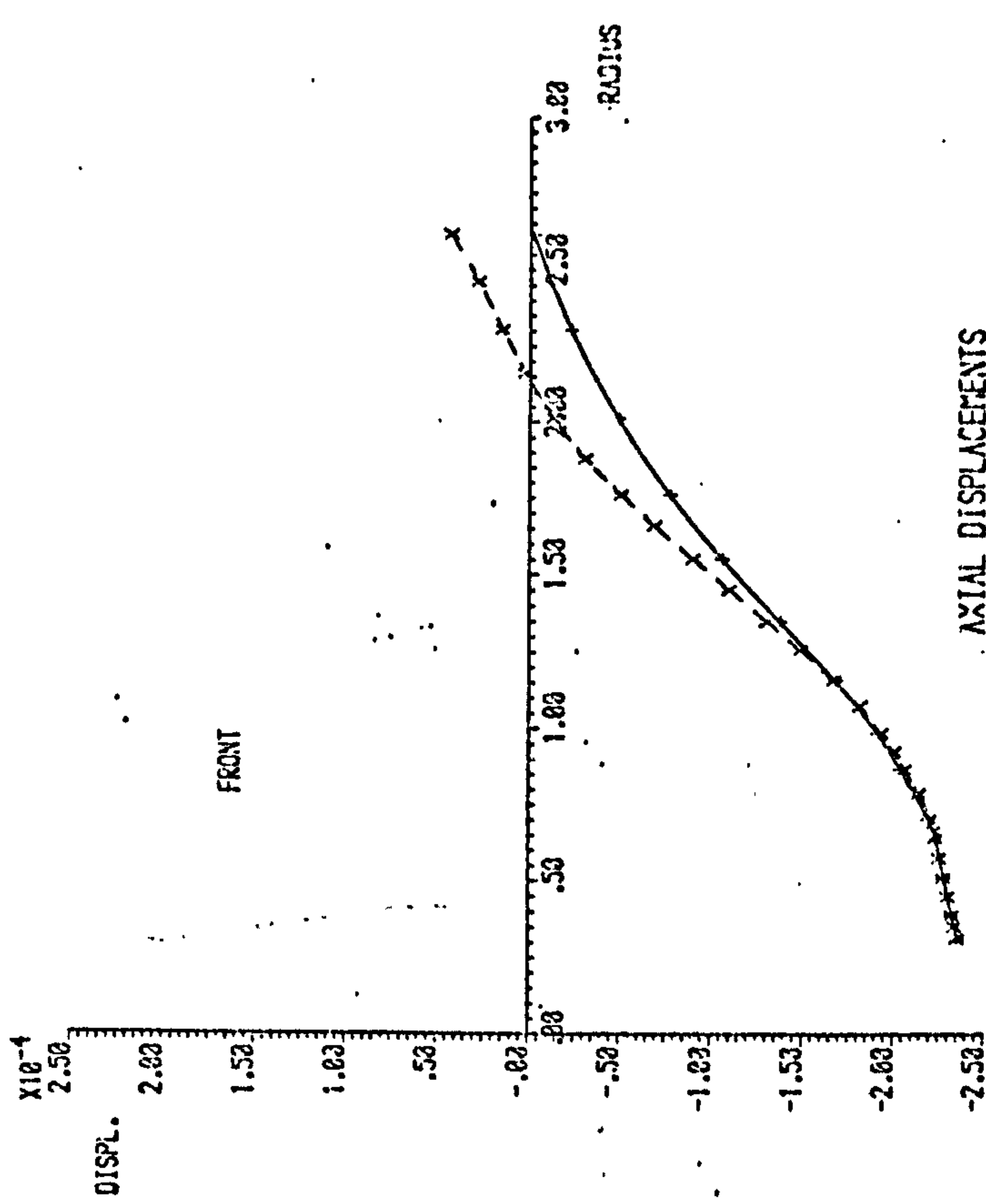


AXIAL DISPLACEMENTS

IMPELLER MODEL - P MESH III 7 BLADES

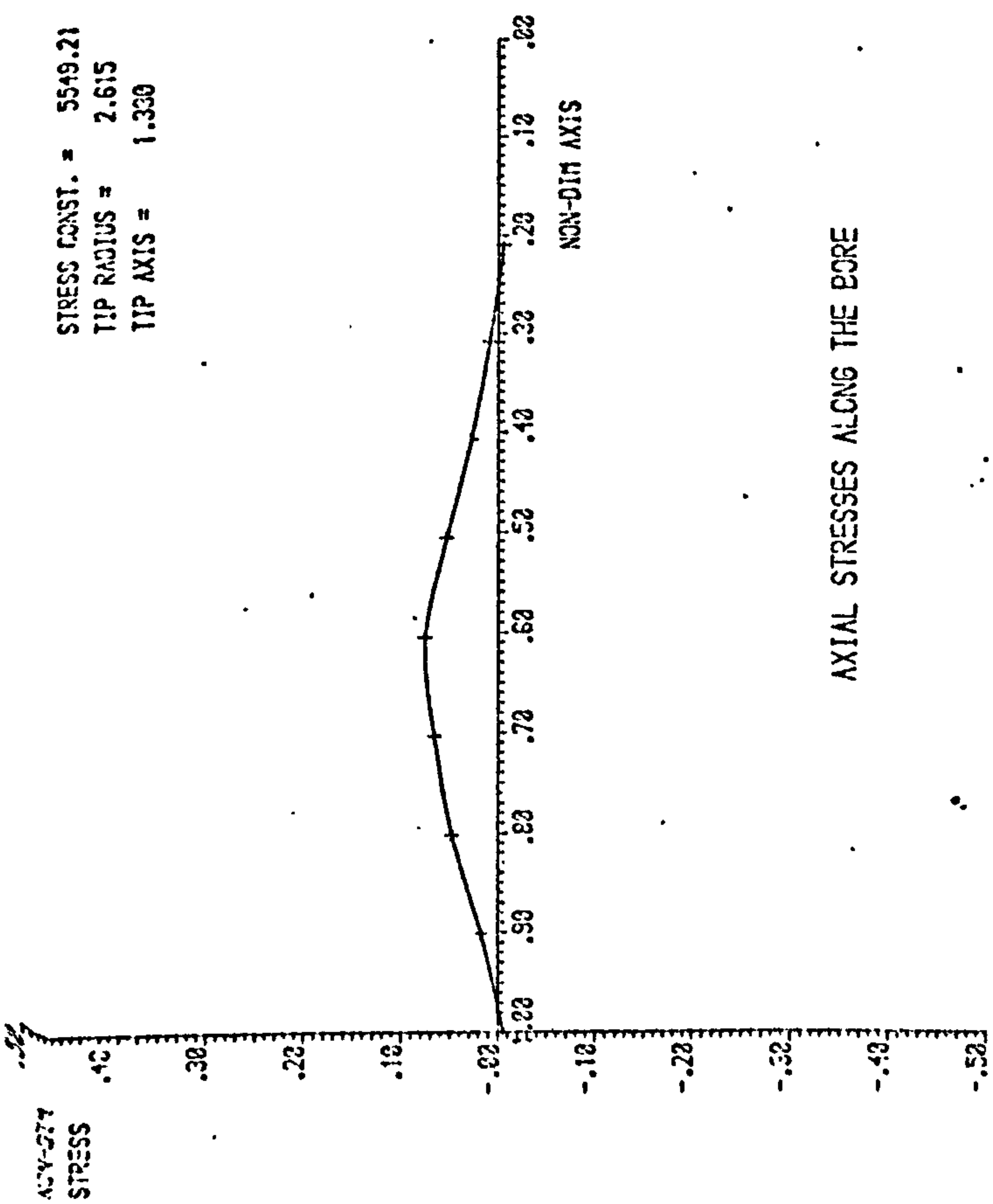
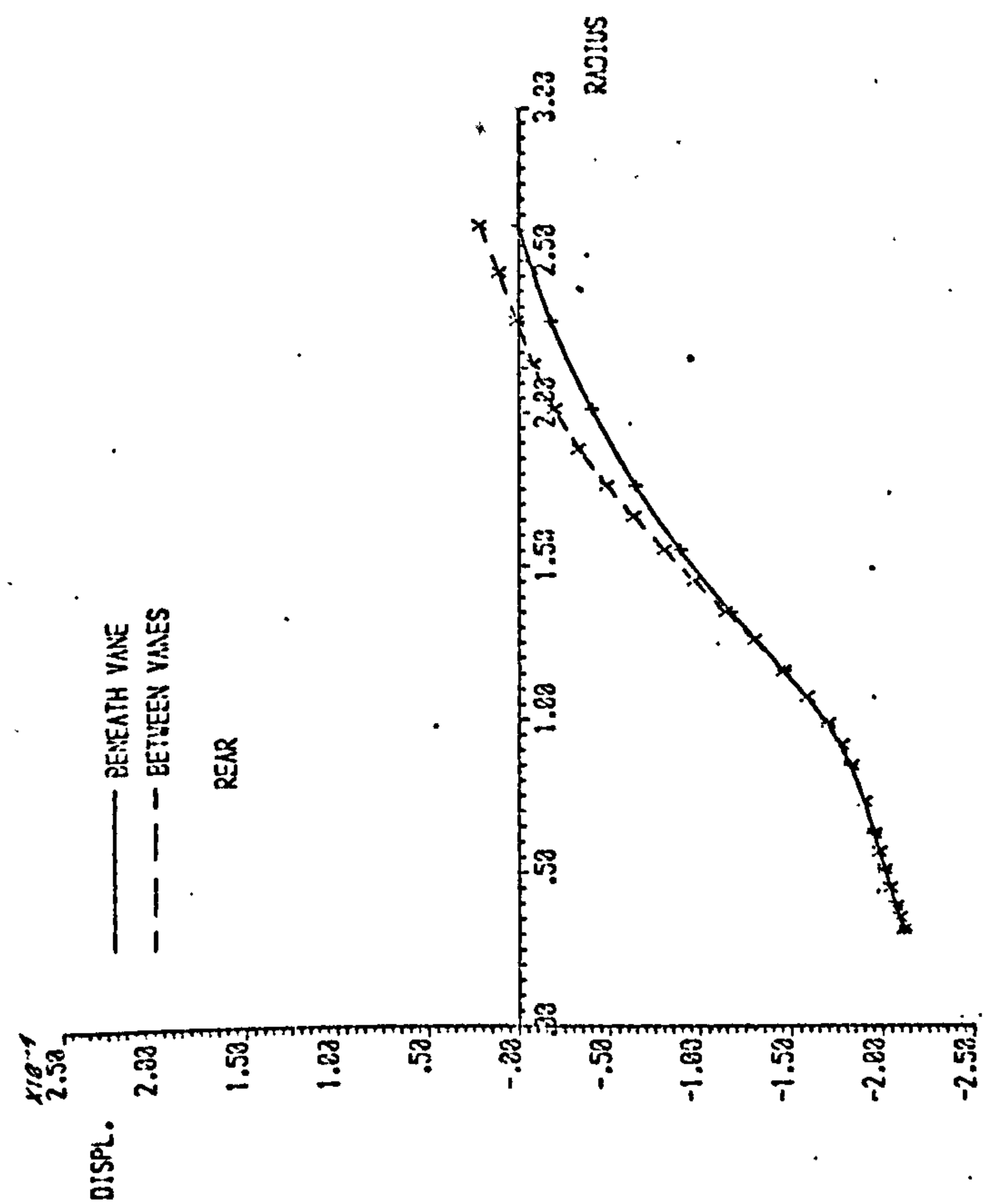
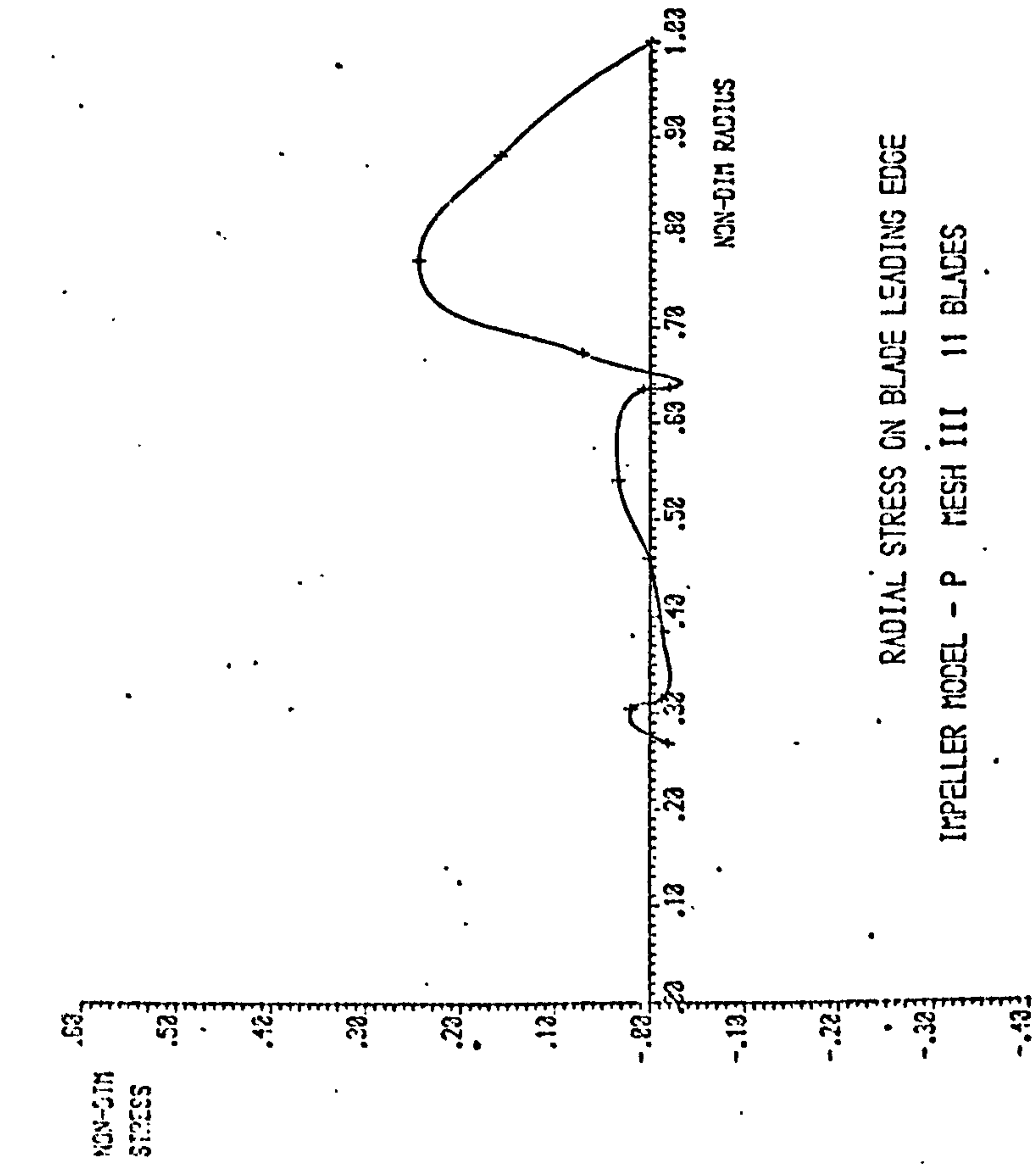
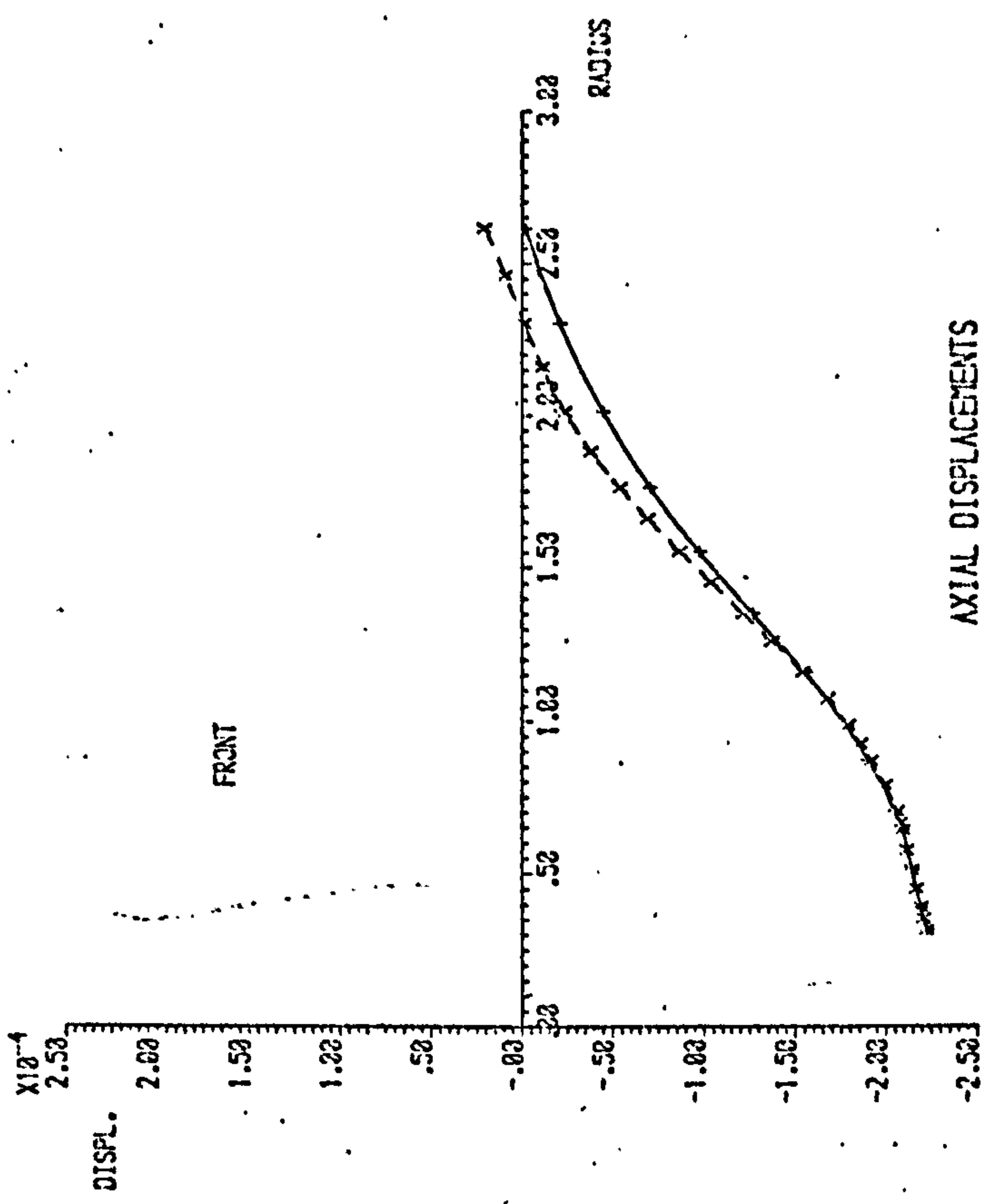
IMPELLER MODEL - P MESH III 7 BLADES

FIG 4-17(e)



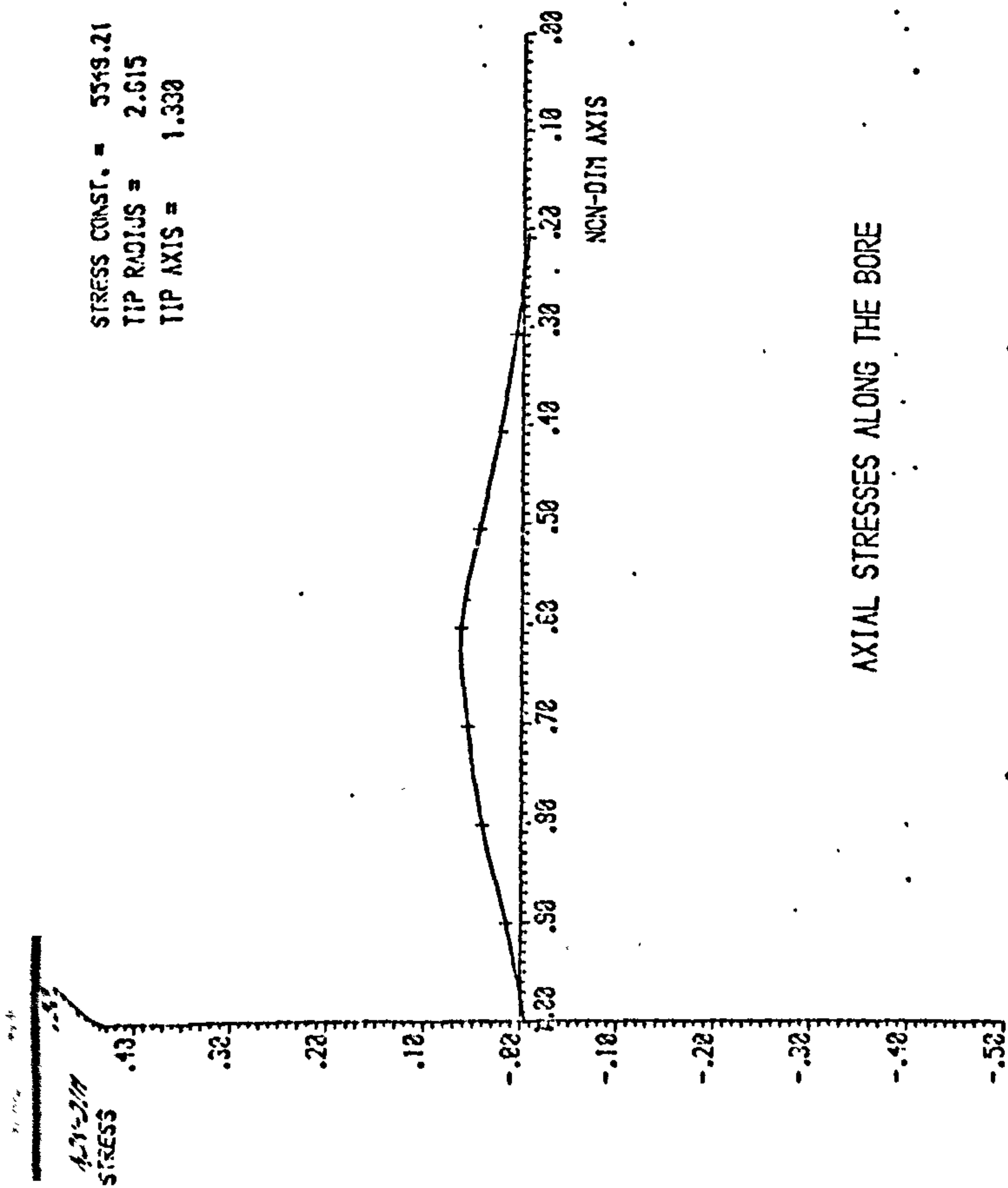
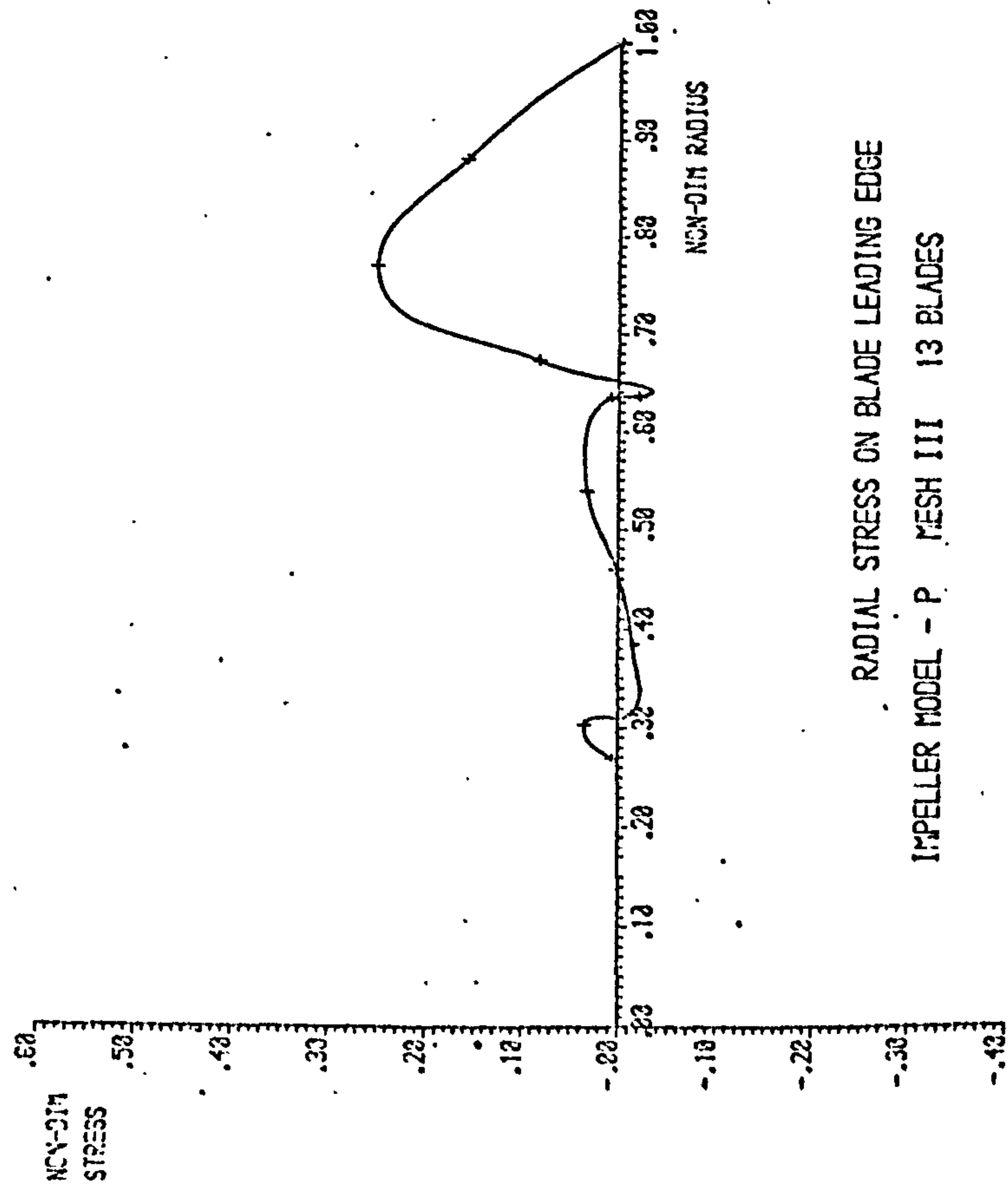
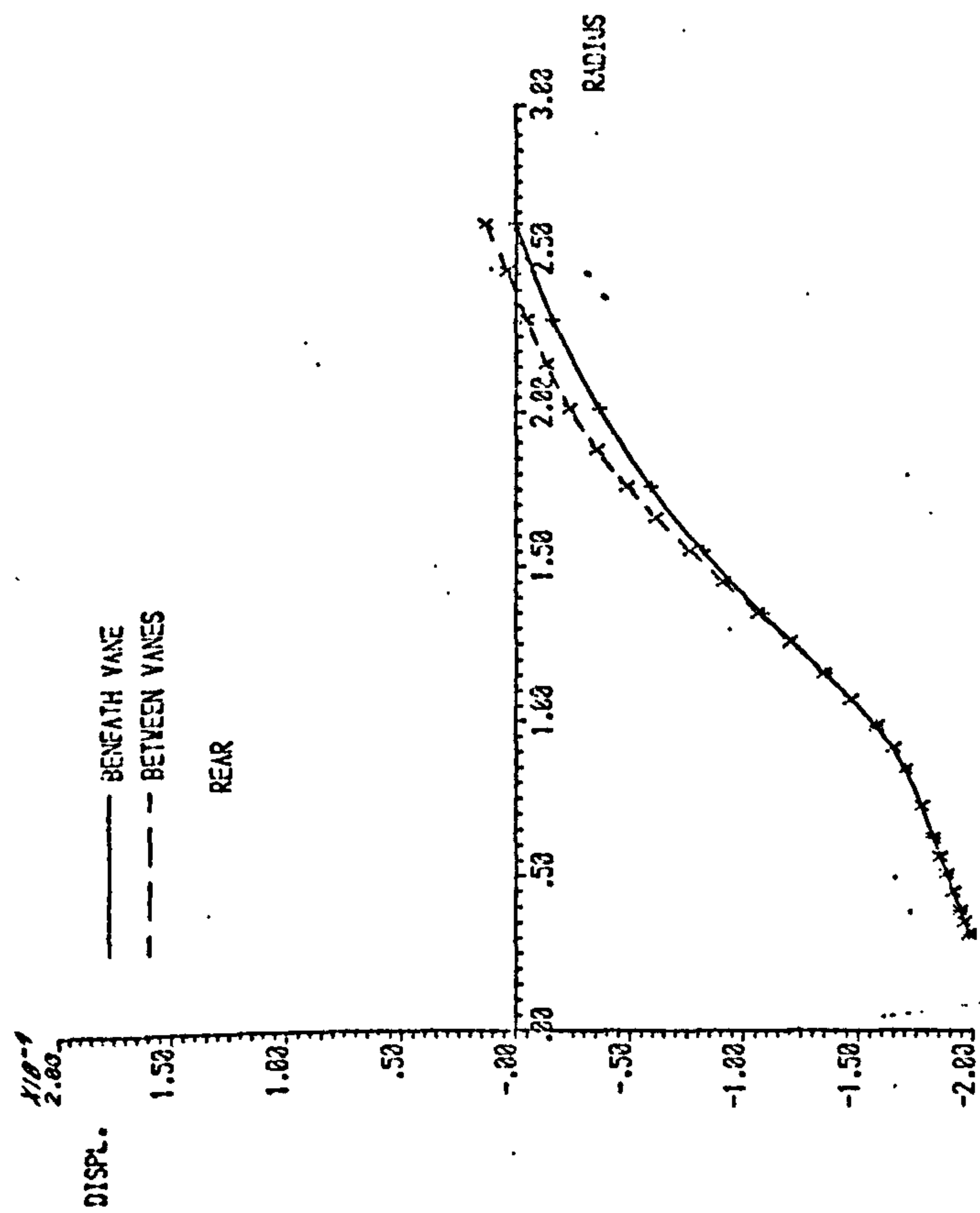
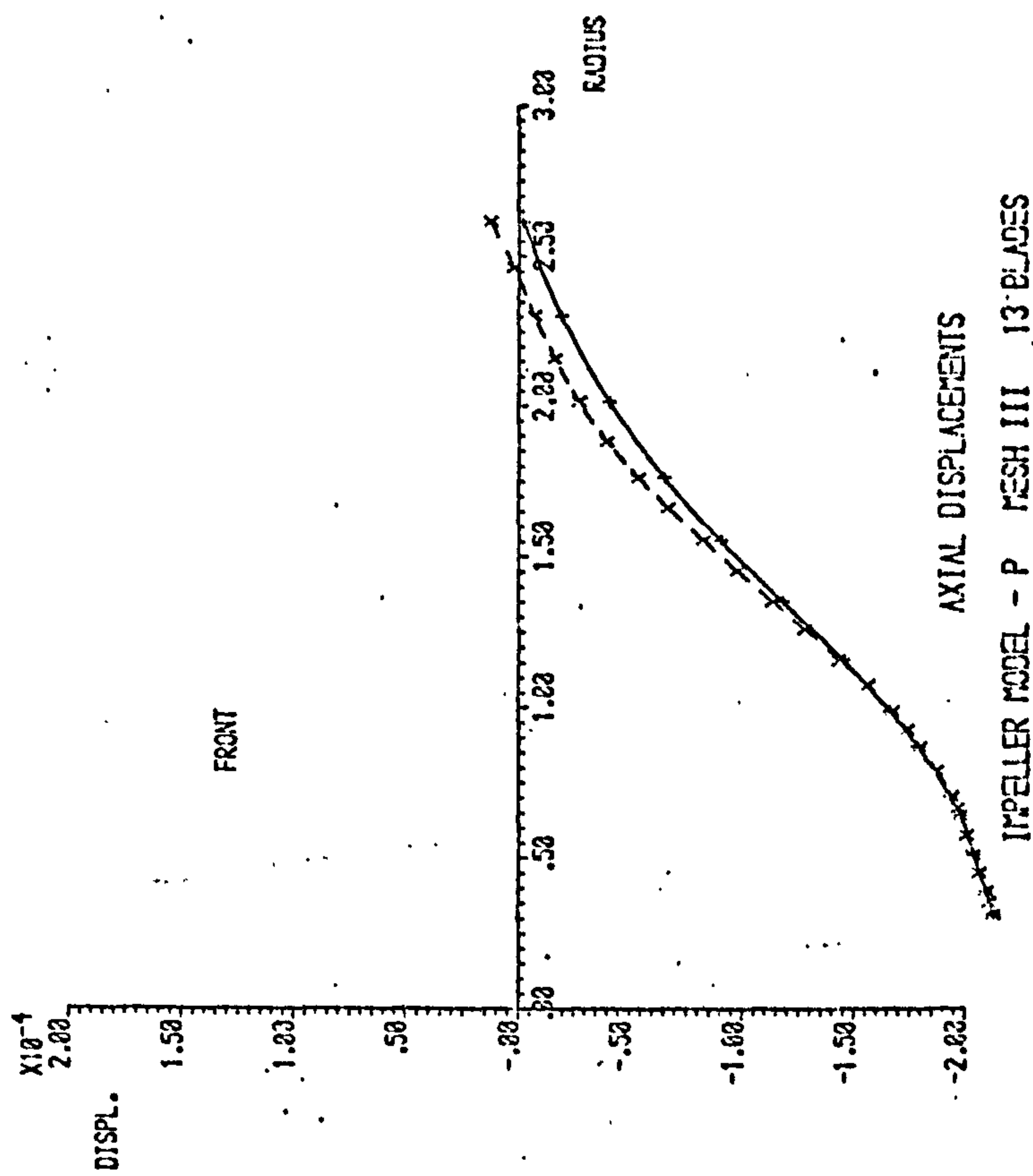
IMPELLER MODEL - P MESH III 9 BLADES

FIG 4-17(f)



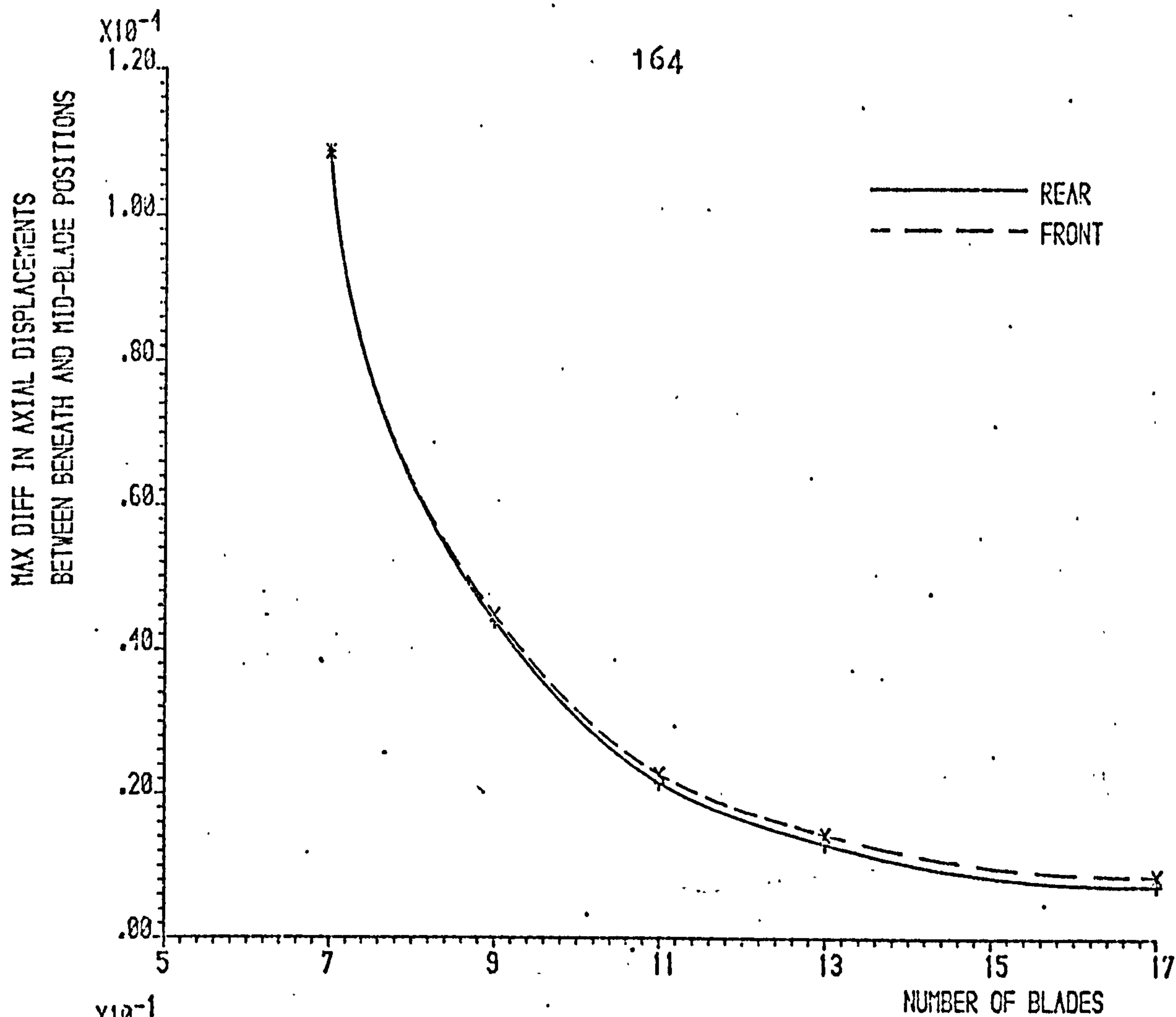
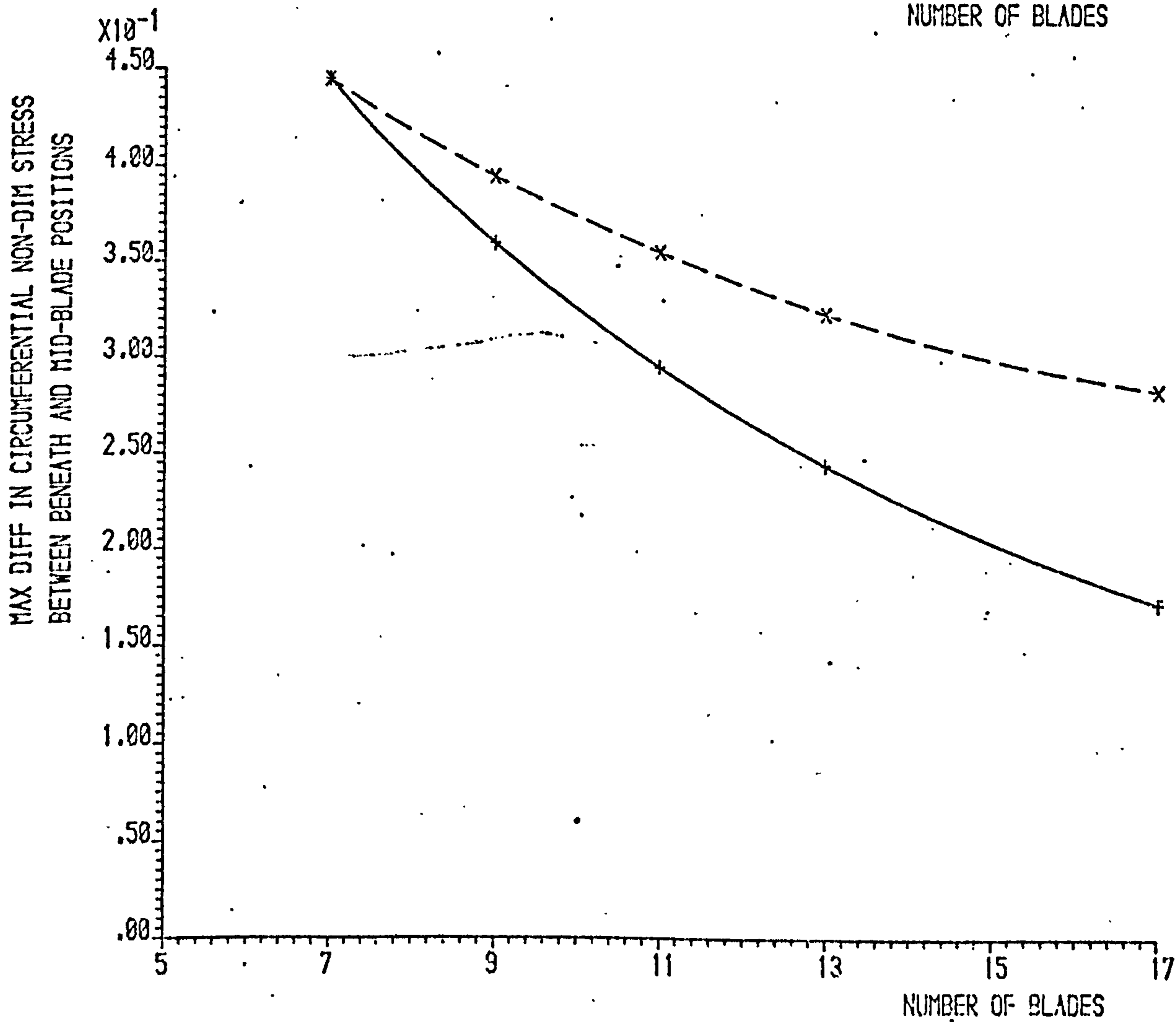
AXIAL DISPLACEMENTS
IMPELLER MODEL - P MESH III 11 BLADES

FIG 4-17(g)



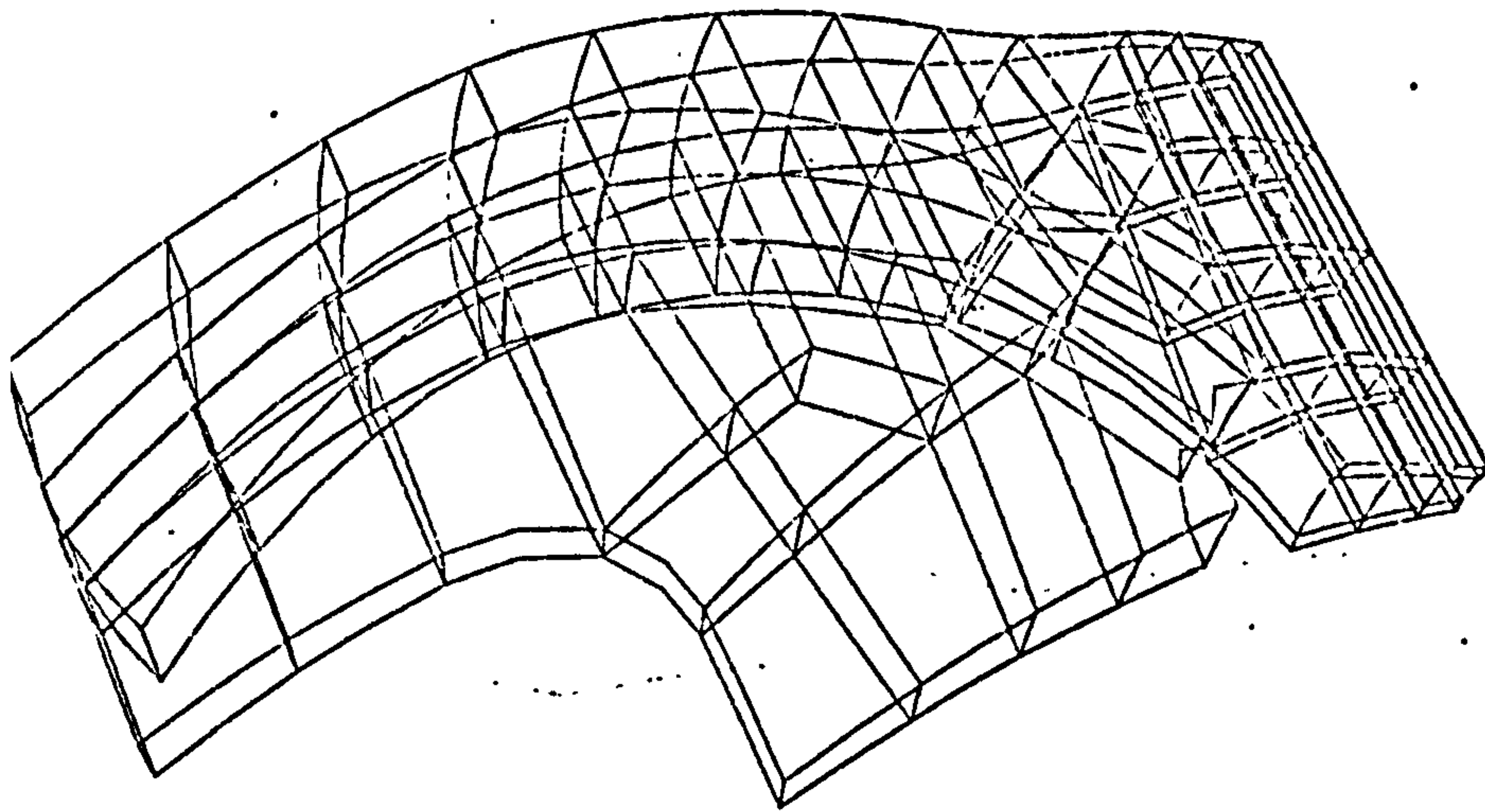
AXIAL DISPLACEMENTS
IMPELLER MODEL - P MESH III 13 BLADES

FIG 4-17(h)



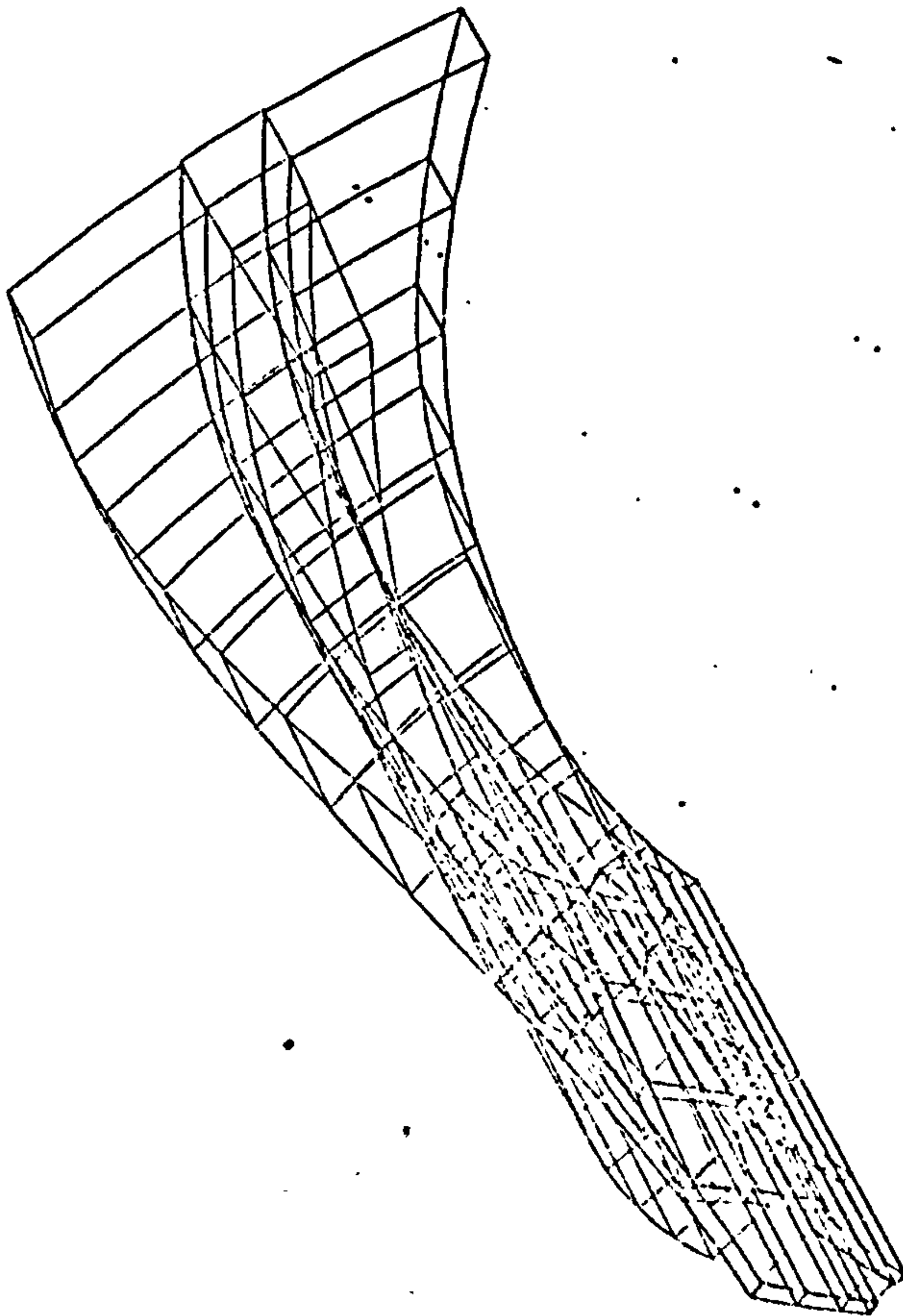
INTER-BLADE BENDING EFFECT IN IMPELLER MODEL - P

FIG 4-18



3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN IN SCALE: 55.00 TIMES THE SUPPLIED UNITS

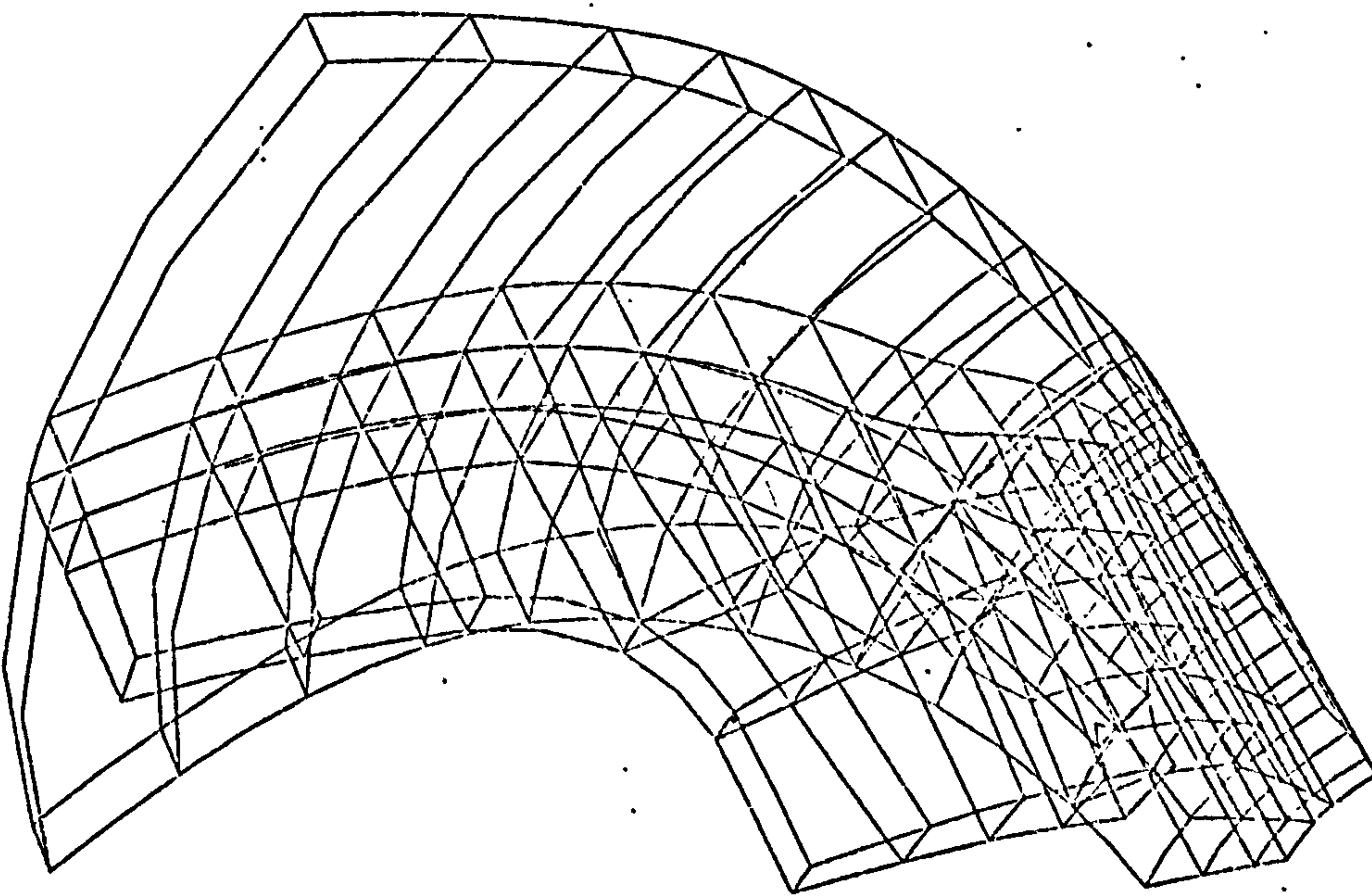
IMPELLER MODEL-P MESH III 17 BLADES CUR. 45 DEG.



3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN IN SCALE: 55.00 TIMES THE SUPPLIED UNITS

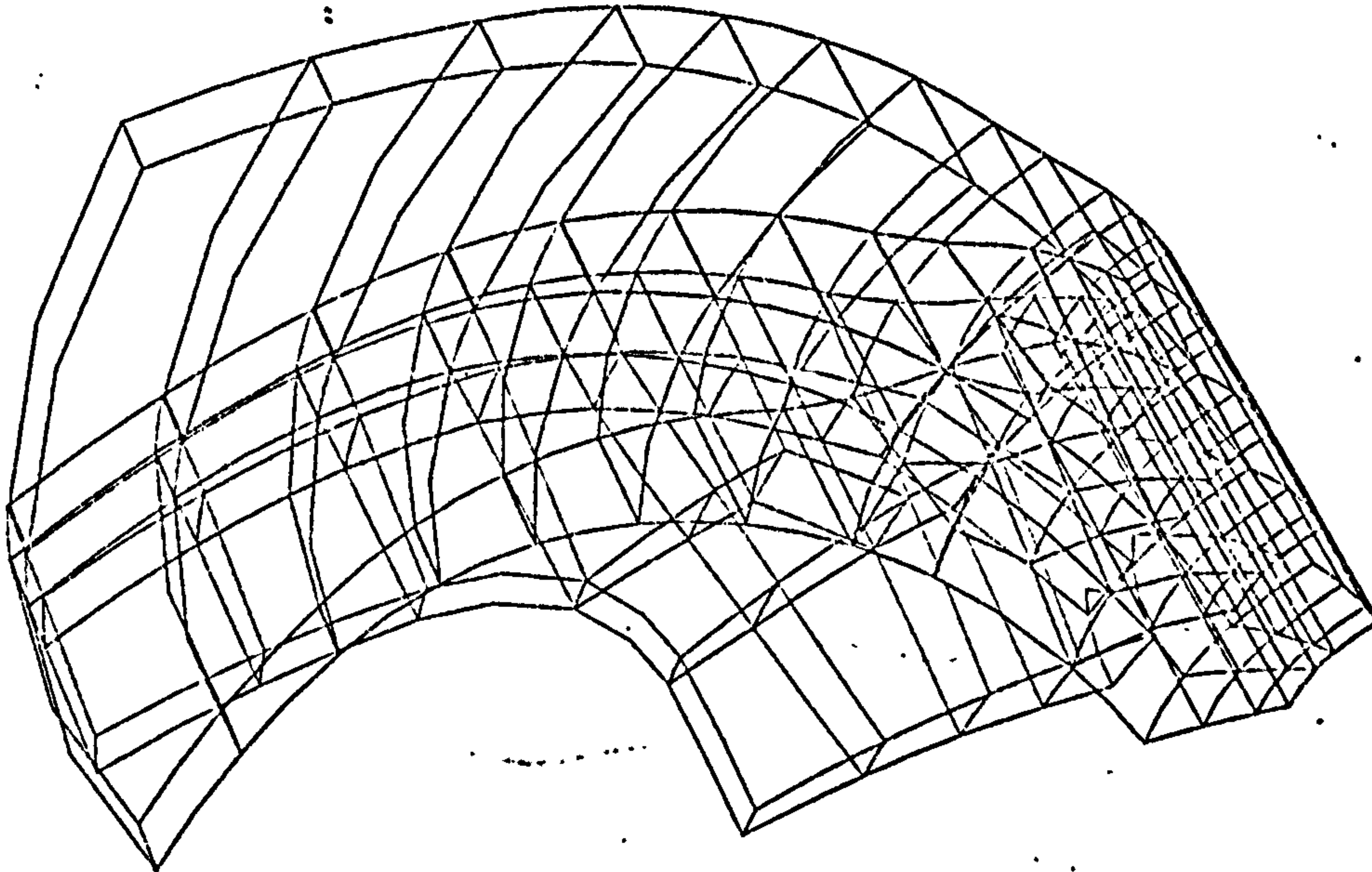
IMPELLER MODEL-P MESH III 17 BLADES CUR. 30 DEG.

FIG 4-19(a)



3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS

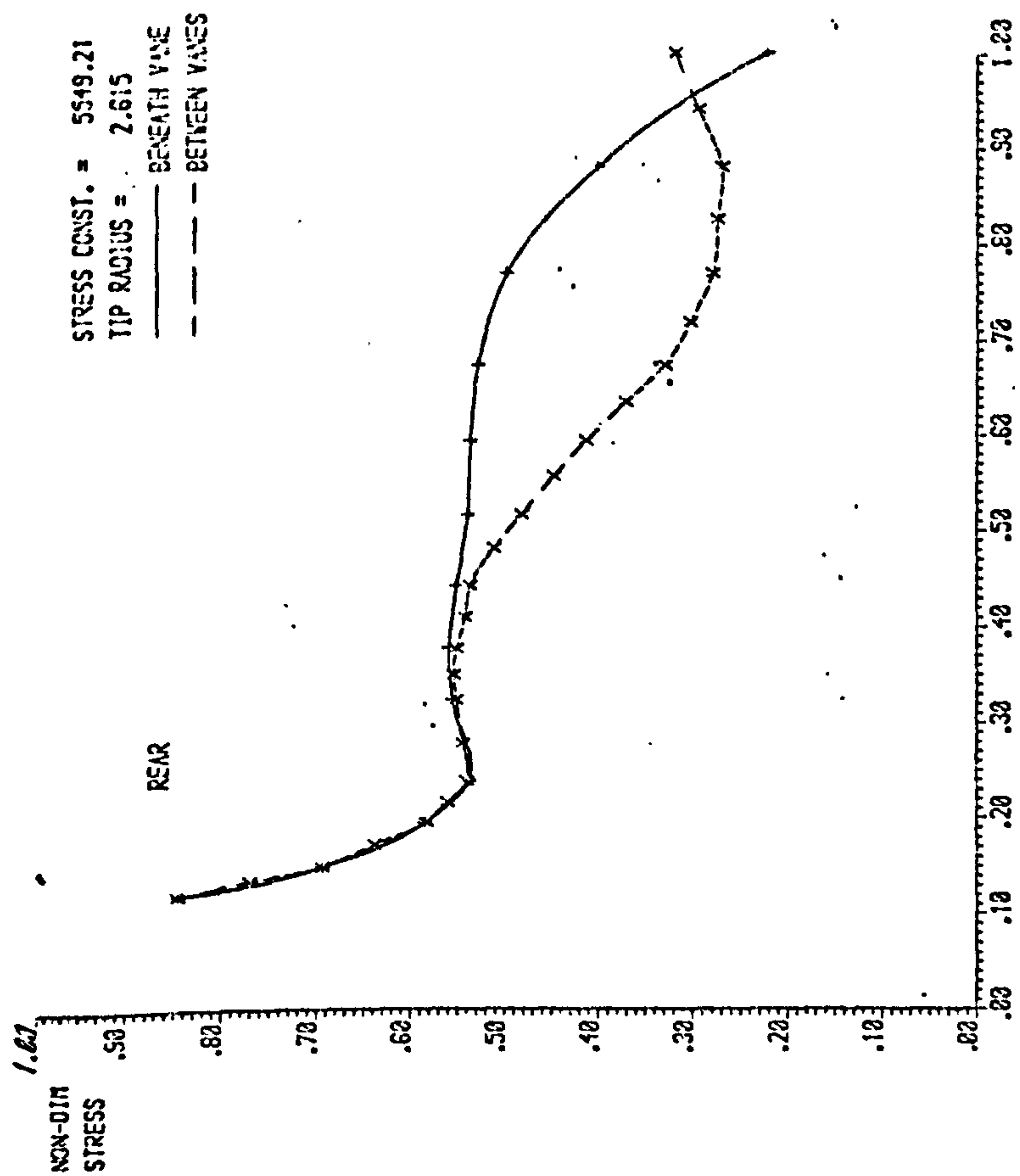
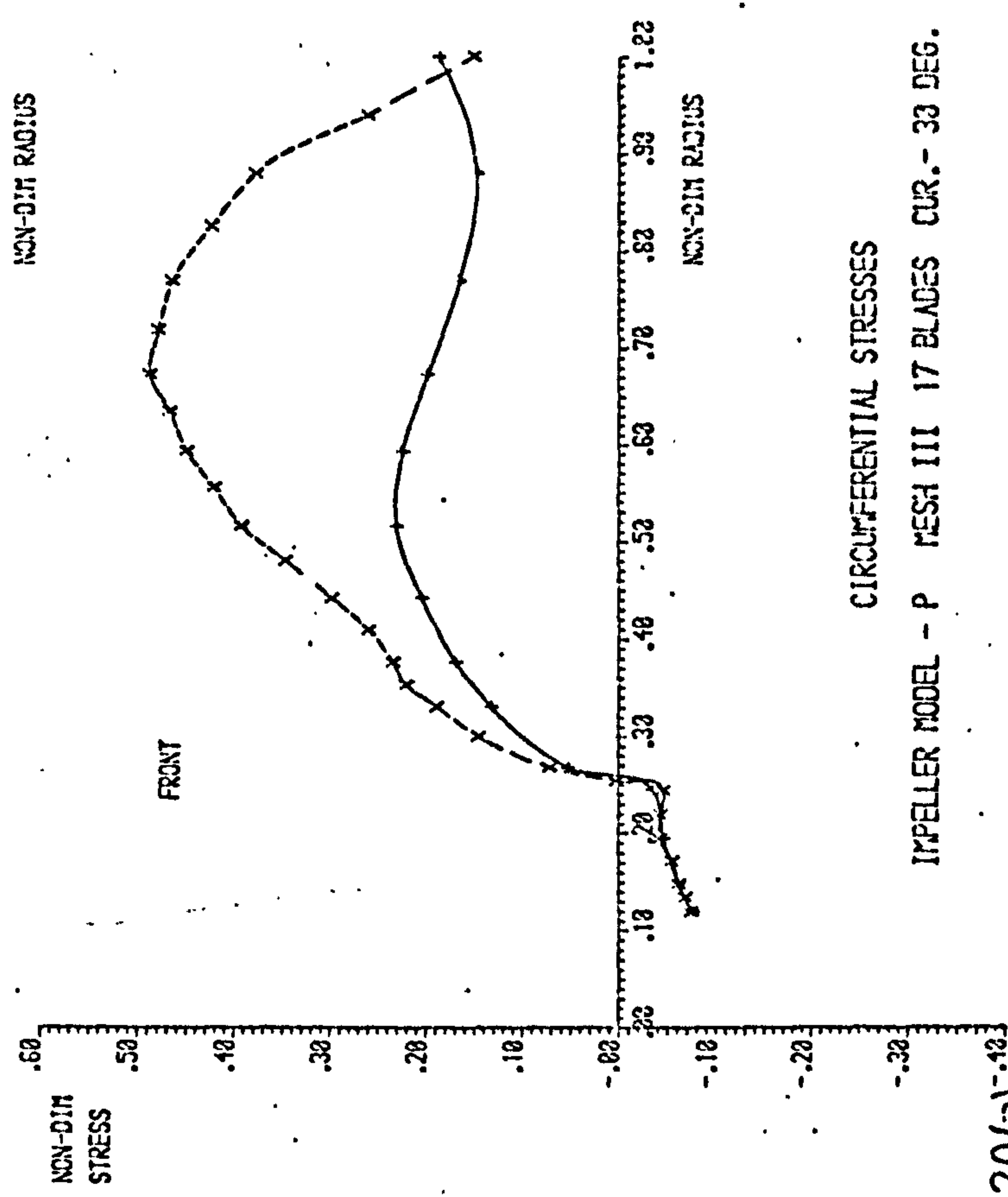
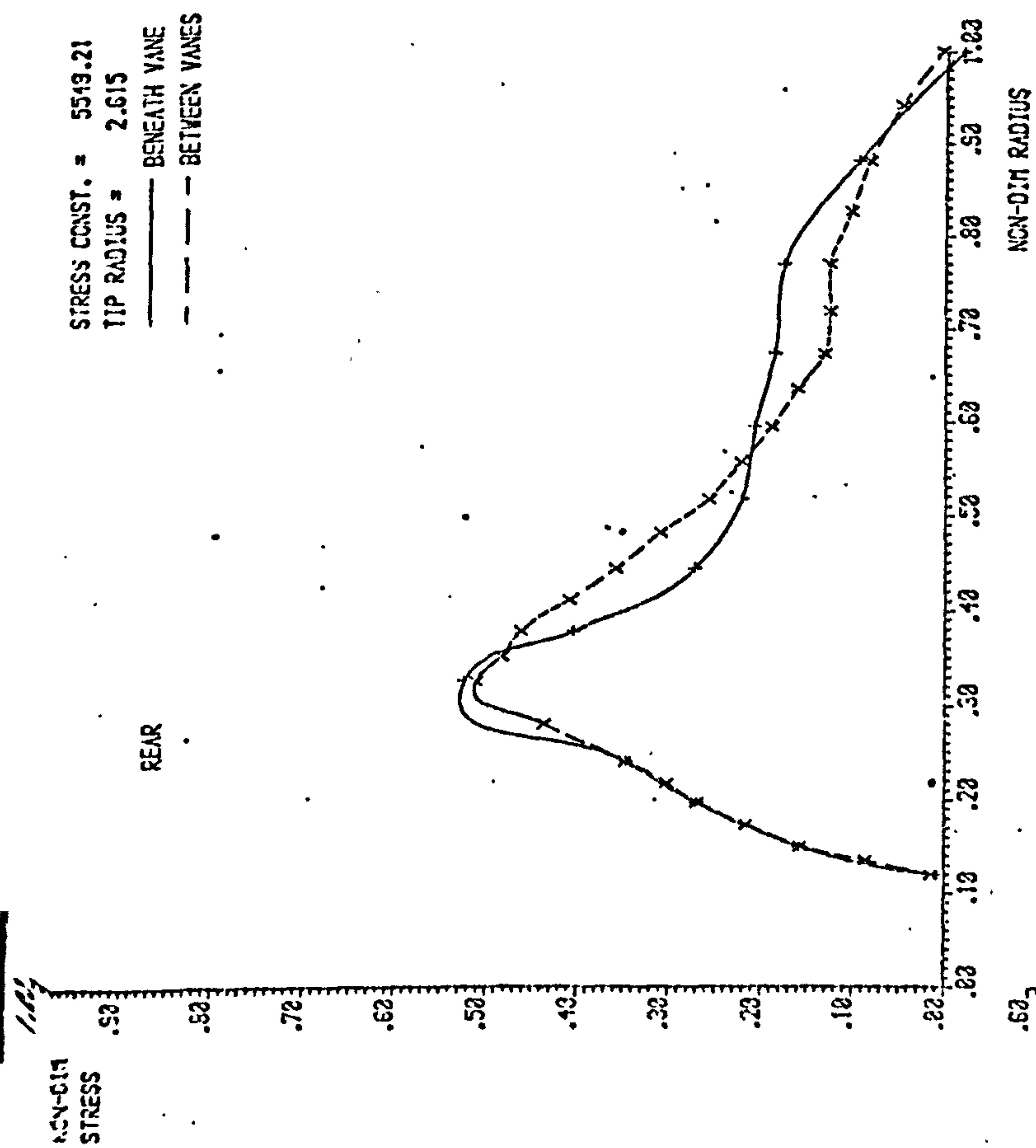
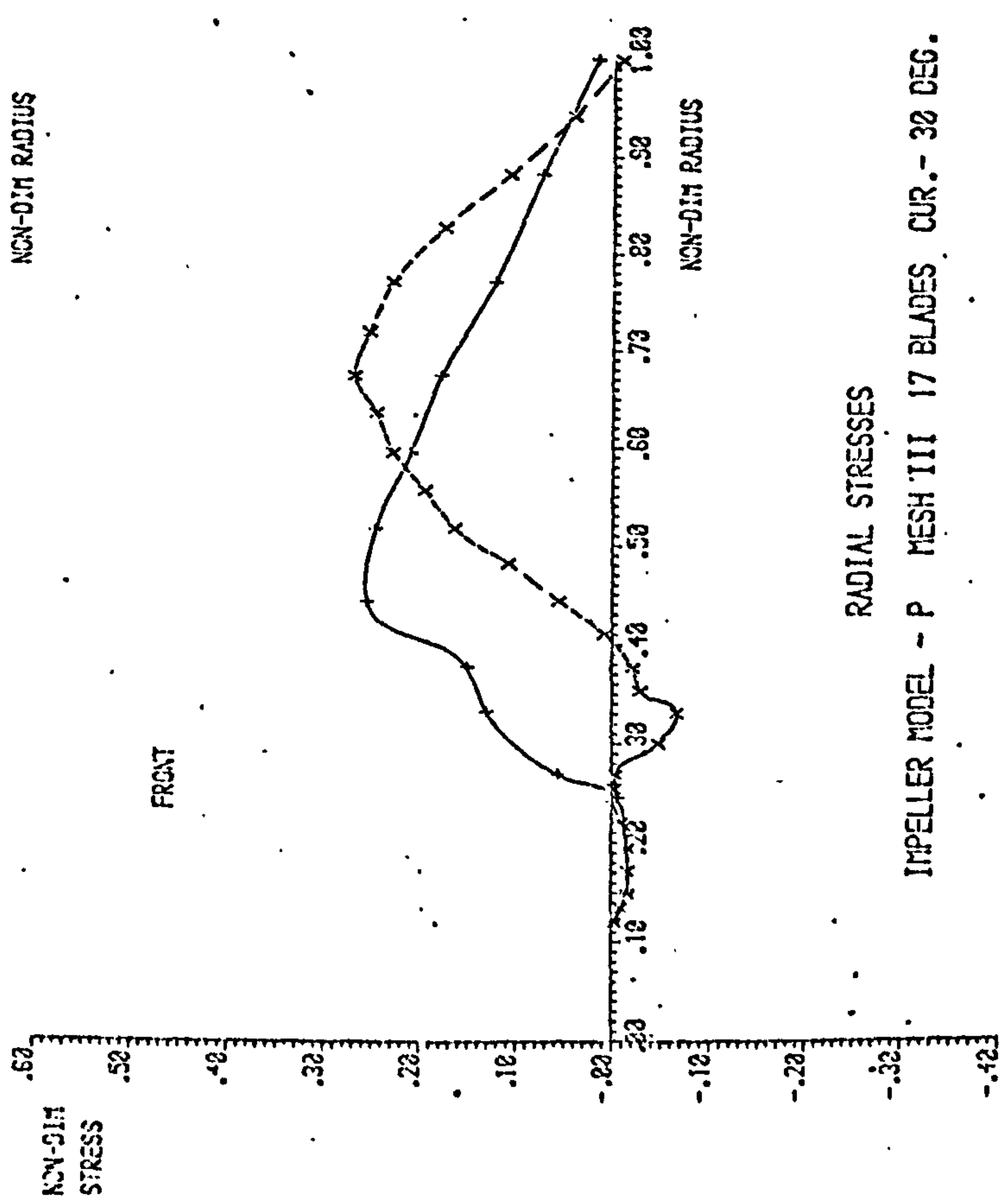
IMPELLER MODEL-P MESH III 7 BLADES CUR. 30 DEG.



3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-P MESH III 7 BLADES CUR. 45 DEG.

FIG 4.19(b)



RADIAL STRESSES

IMPELLER MODEL - P MESH III 17 BLADES CUR. - 30 DEG.

CIRCUMFERENTIAL STRESSES

IMPELLER MODEL - P MESH III 17 BLADES CUR. - 33 DEG.

FIG 4-20(a)

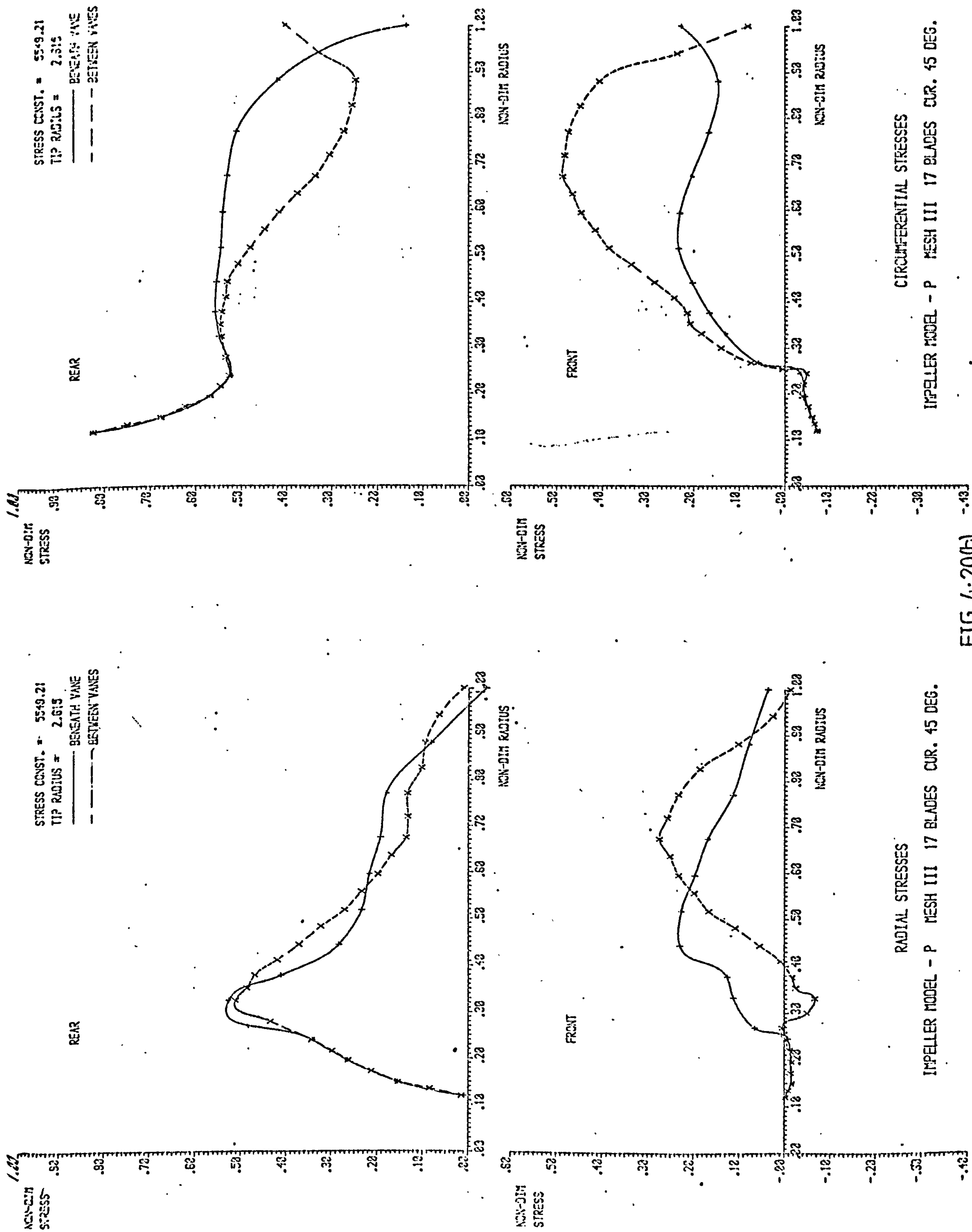
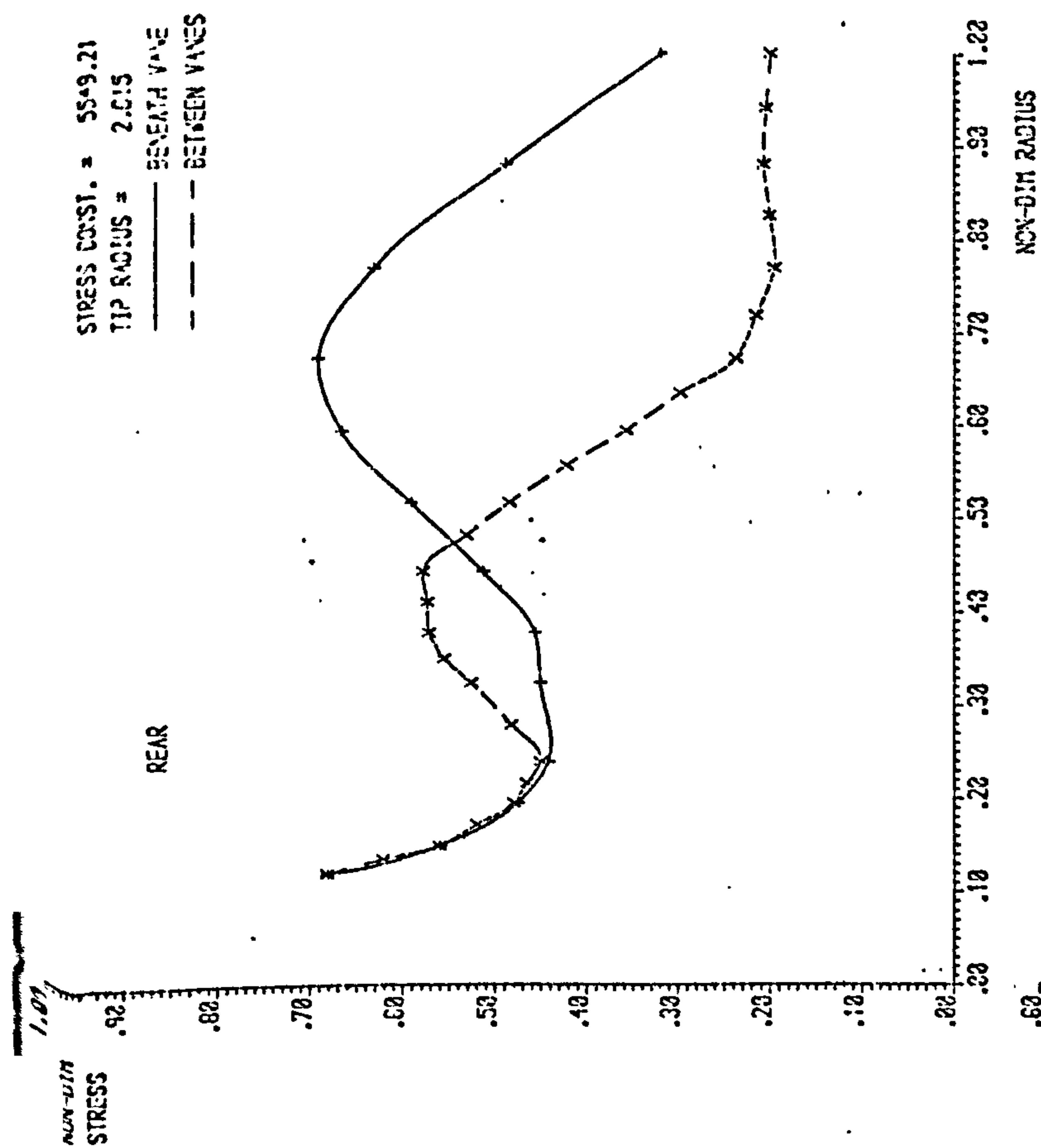
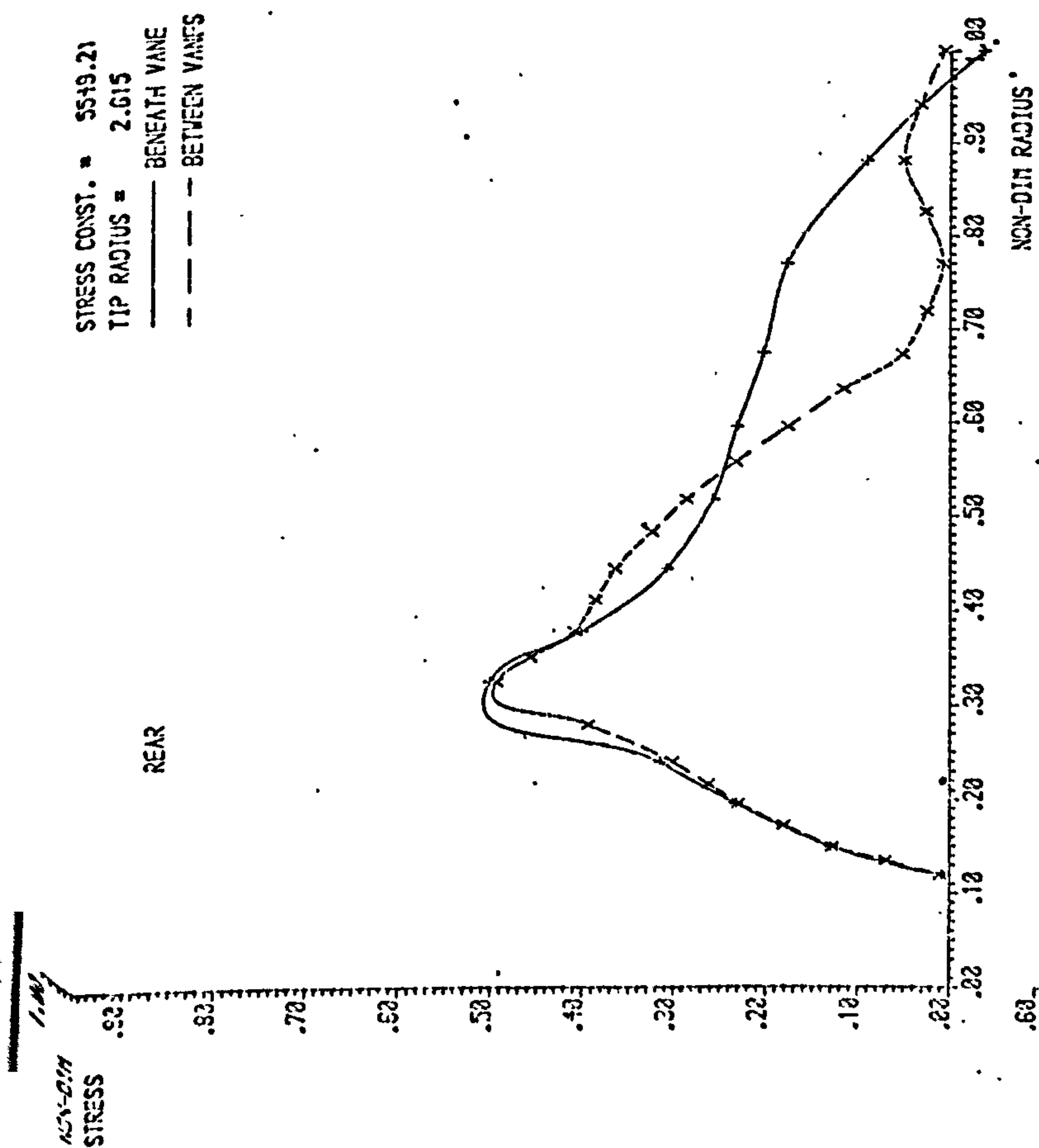
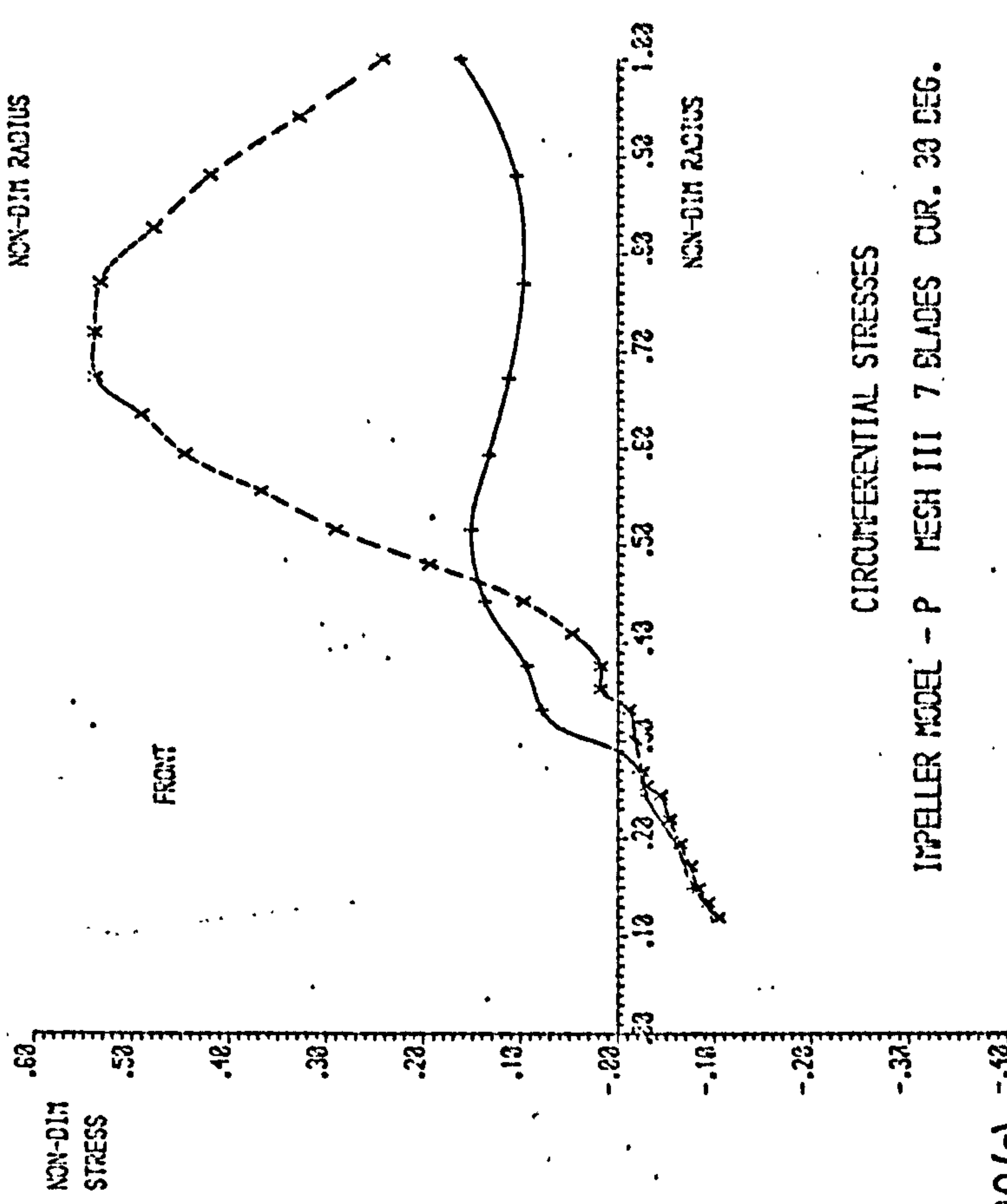
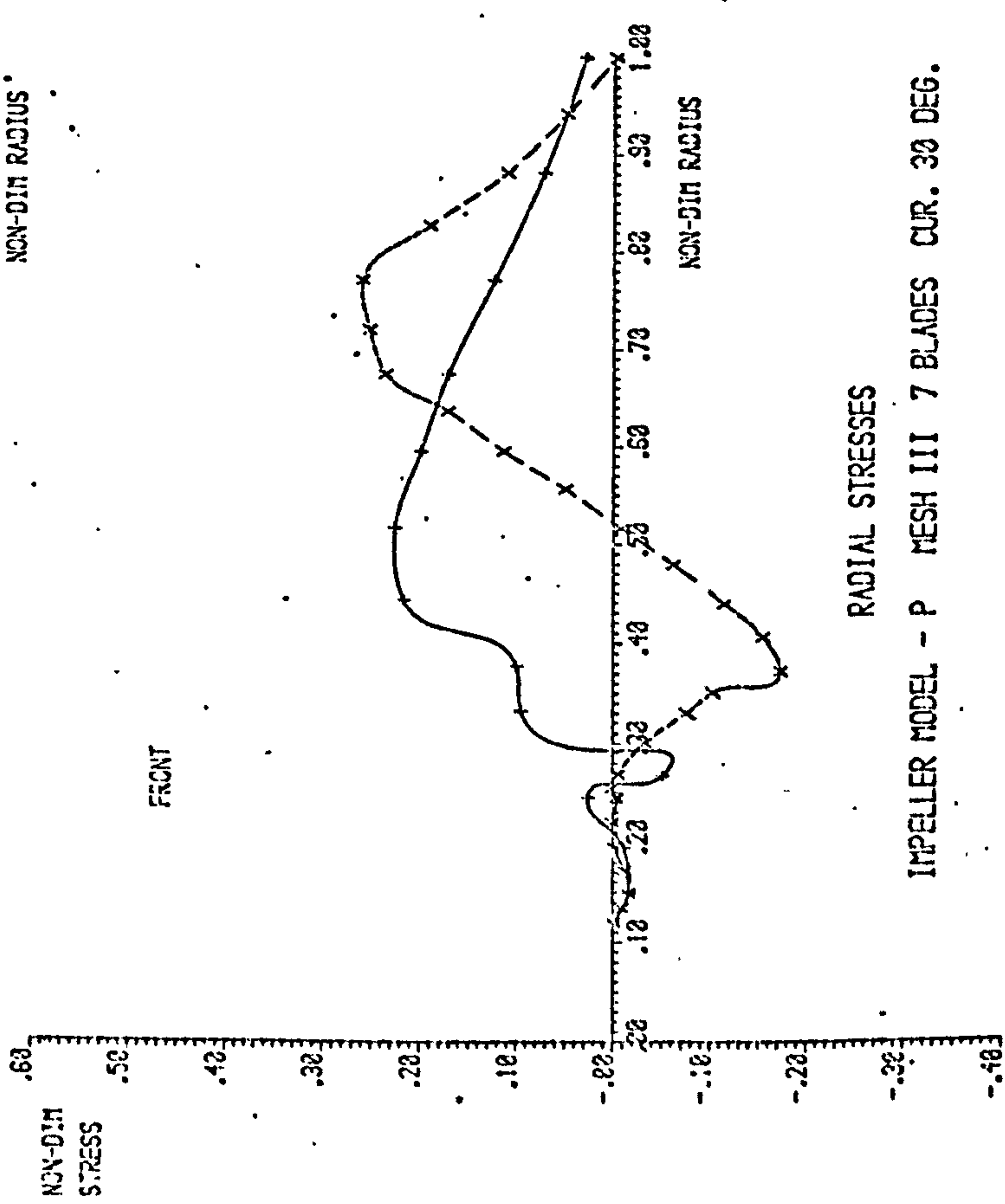


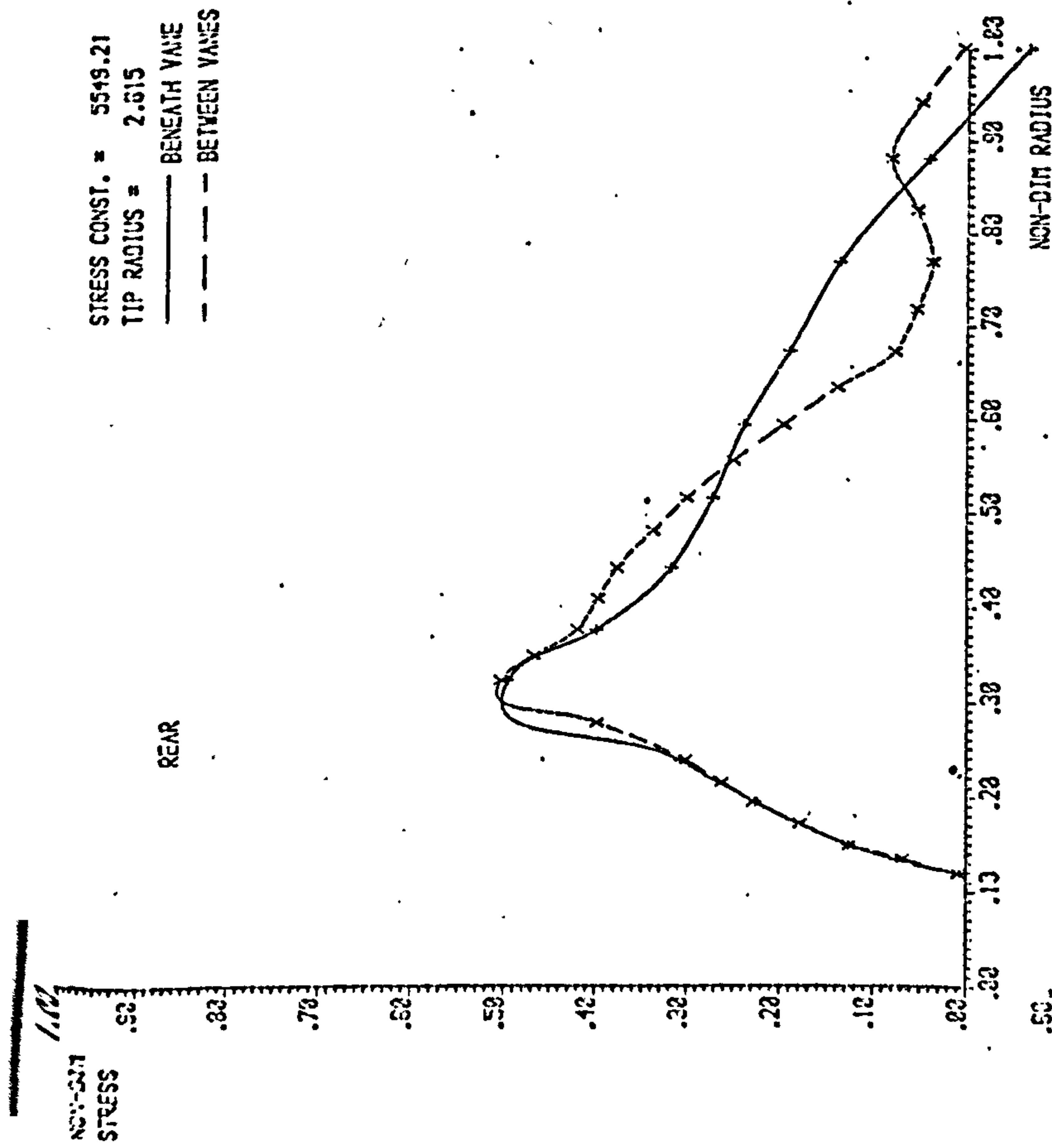
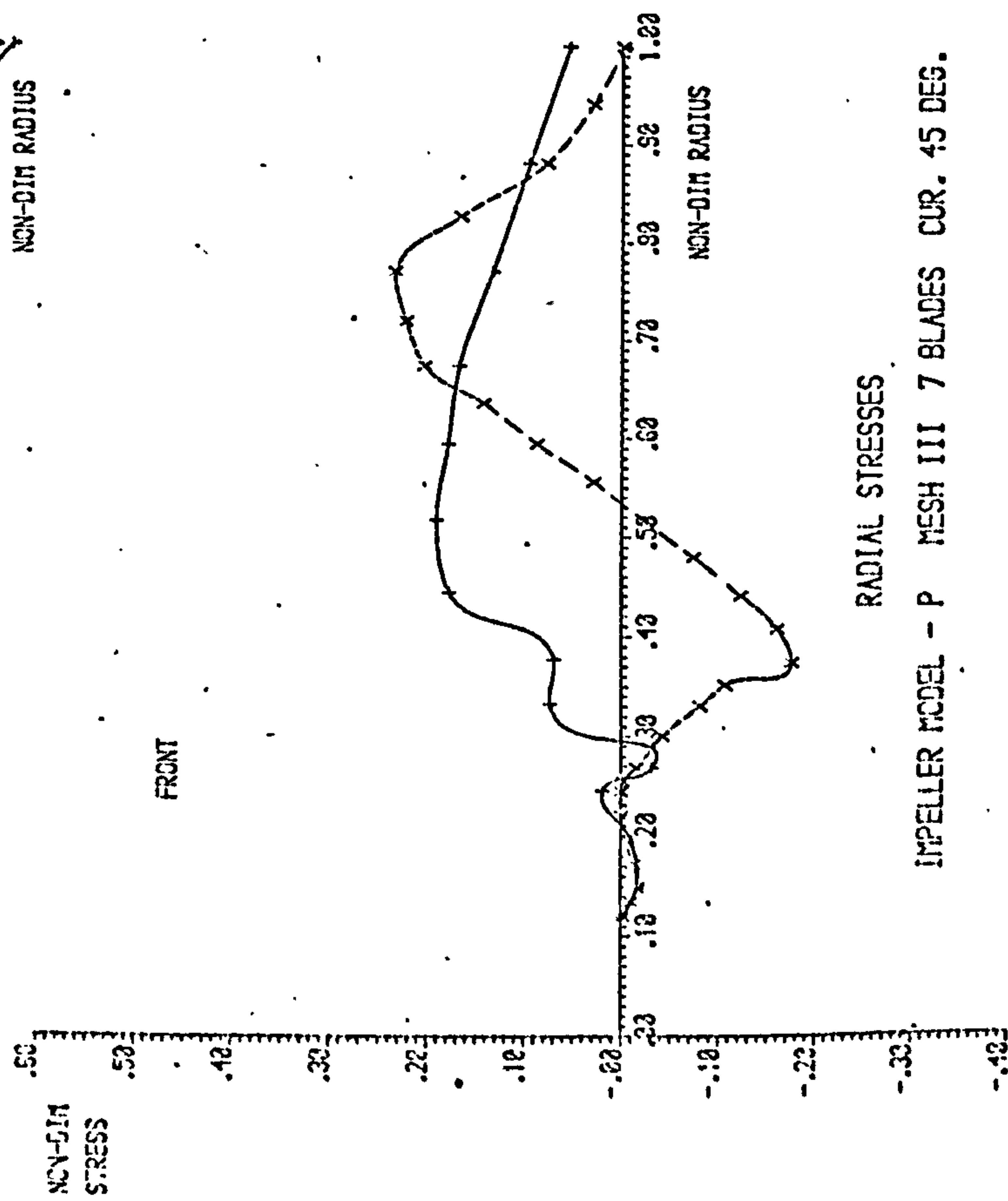
FIG 4-20(b)



CIRCUMFERENTIAL STRESSES
IMPELLER MODEL - P MESH III 7 BLADES CUR. 30 DEG.

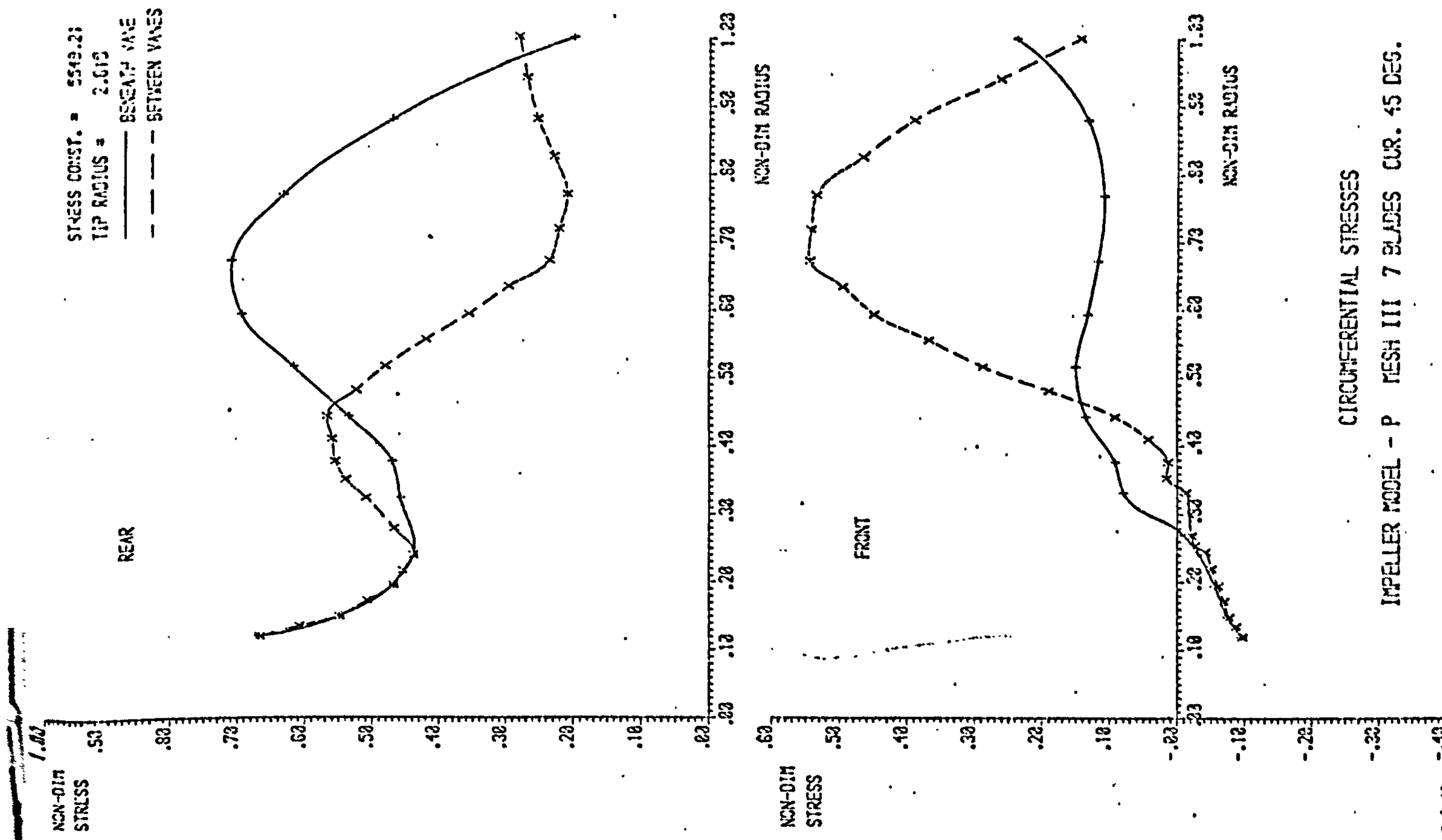
RADIAL STRESSES
IMPELLER MODEL - P MESH III 7 BLADES CUR. 30 DEG.

FIG 4-20(c)



RADIAL STRESSES

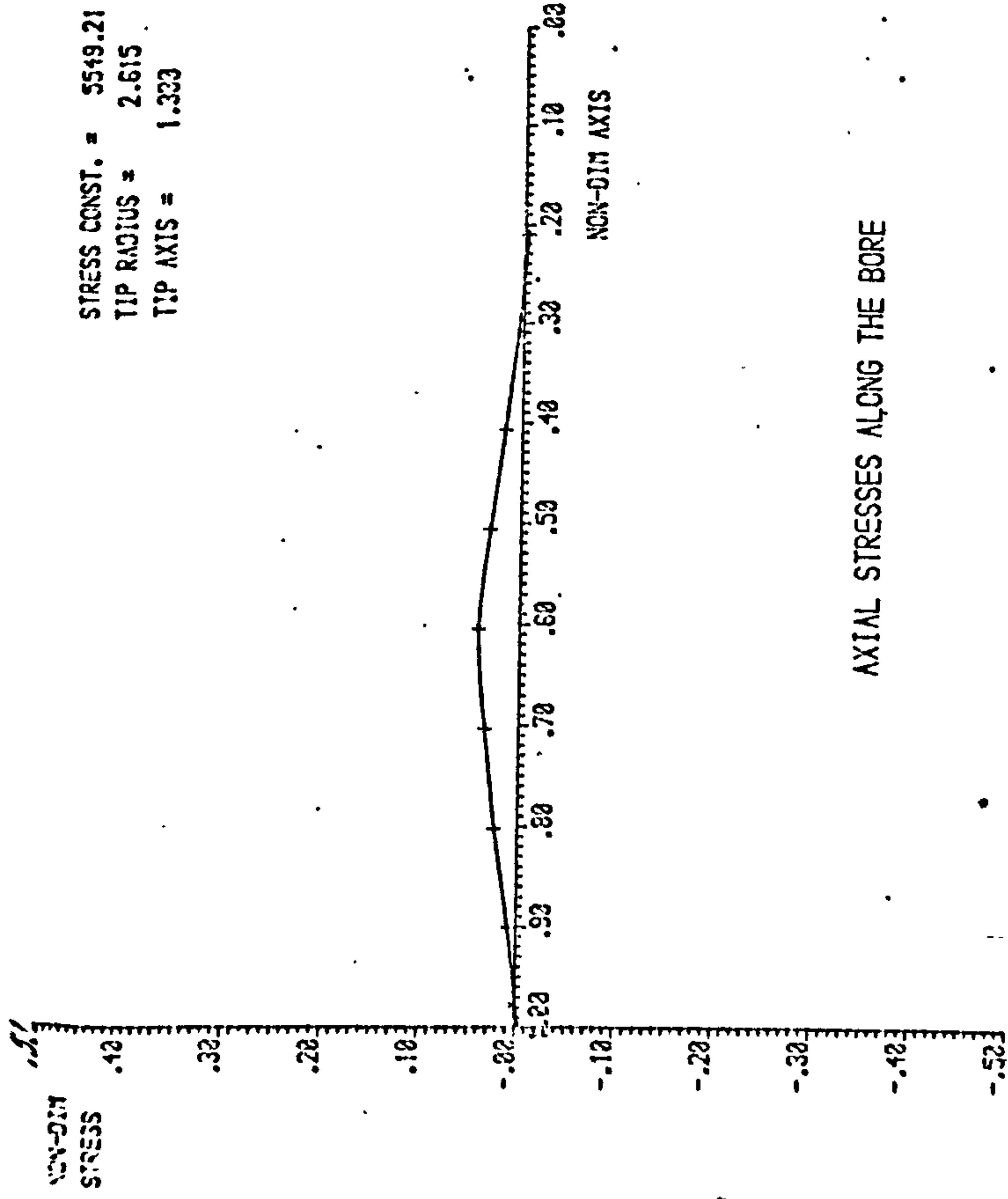
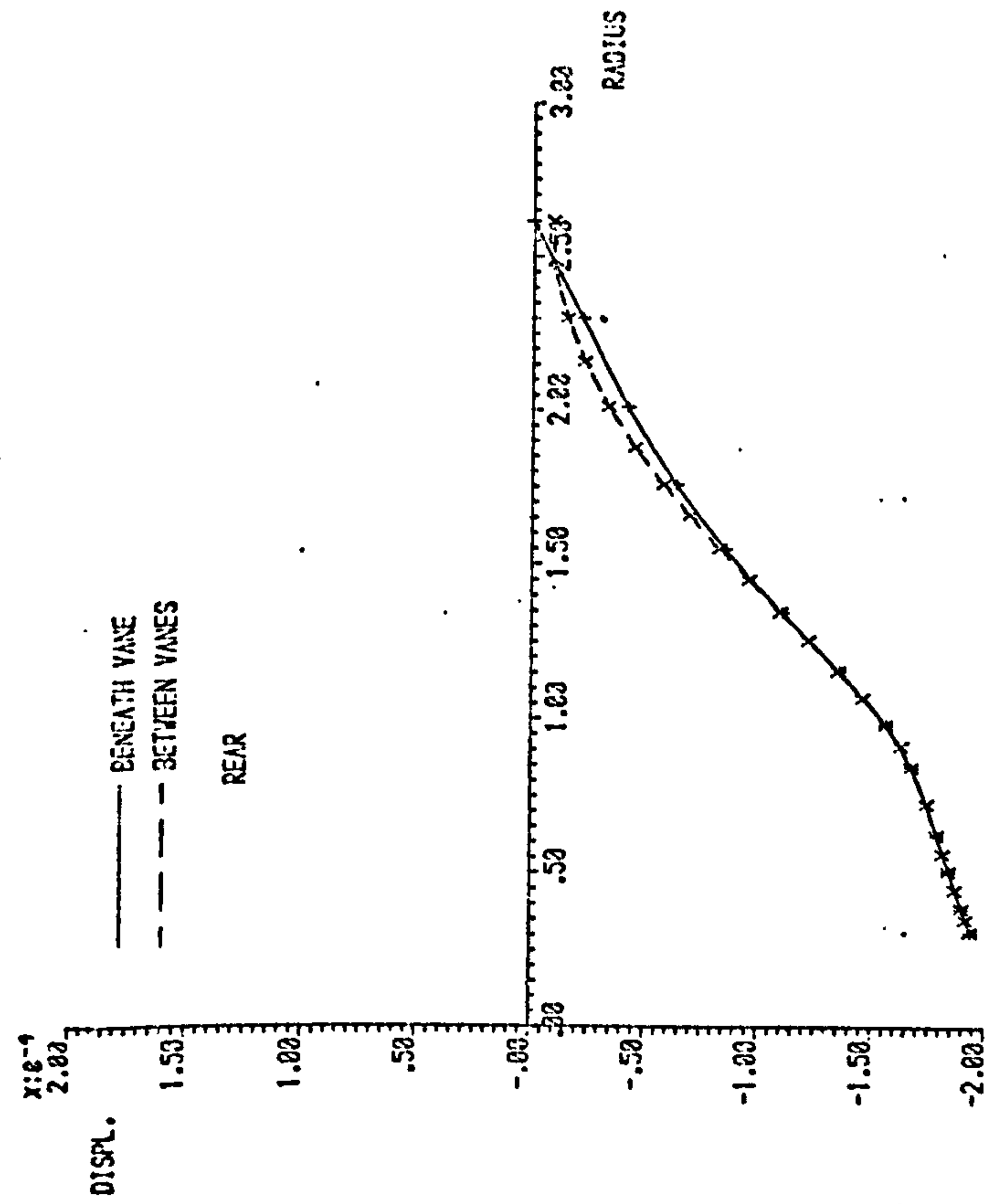
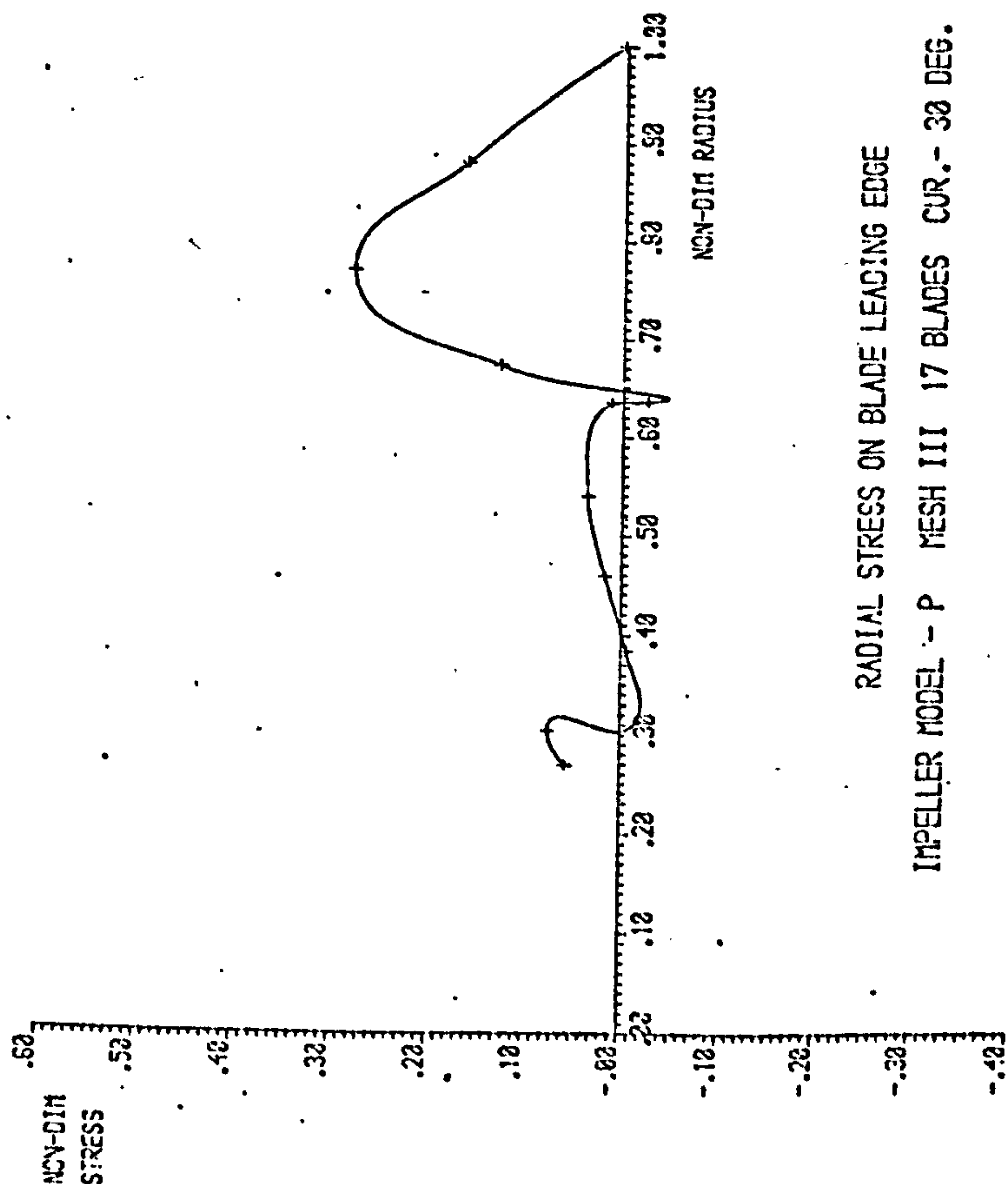
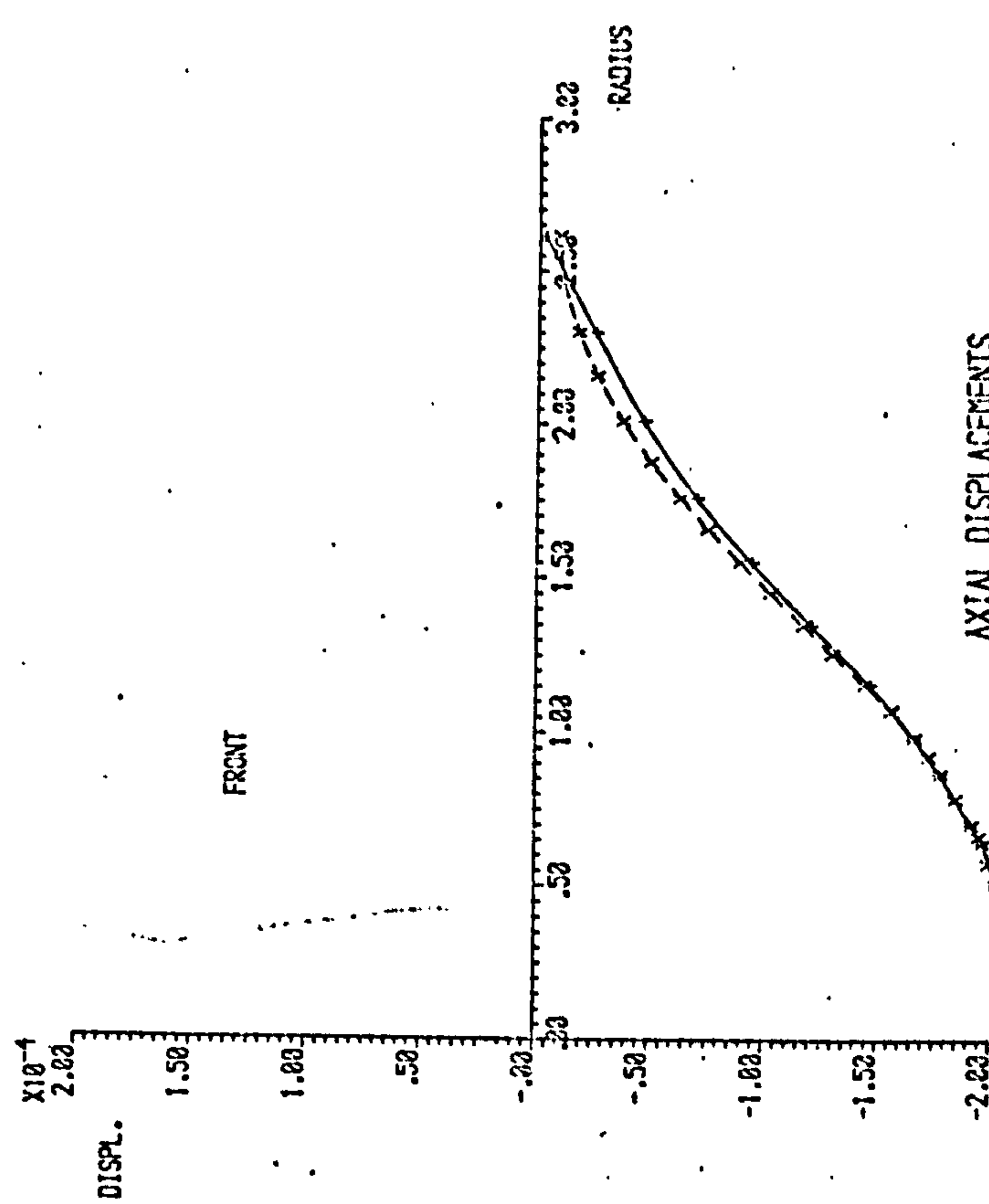
IMPELLER MODEL - P MESH III 7 BLADES CUR. 45 DEG.



CIRCUMFERENTIAL STRESSES

IMPELLER MODEL - P MESH III 7 BLADES CUR. 45 DEG.

FIG 4-20(d)



IMPPELLER MODEL - P MESH III 17 BLADES CUR.- 30 DEG.

AXIAL DISPLACEMENTS

FIG 4-20 (e)

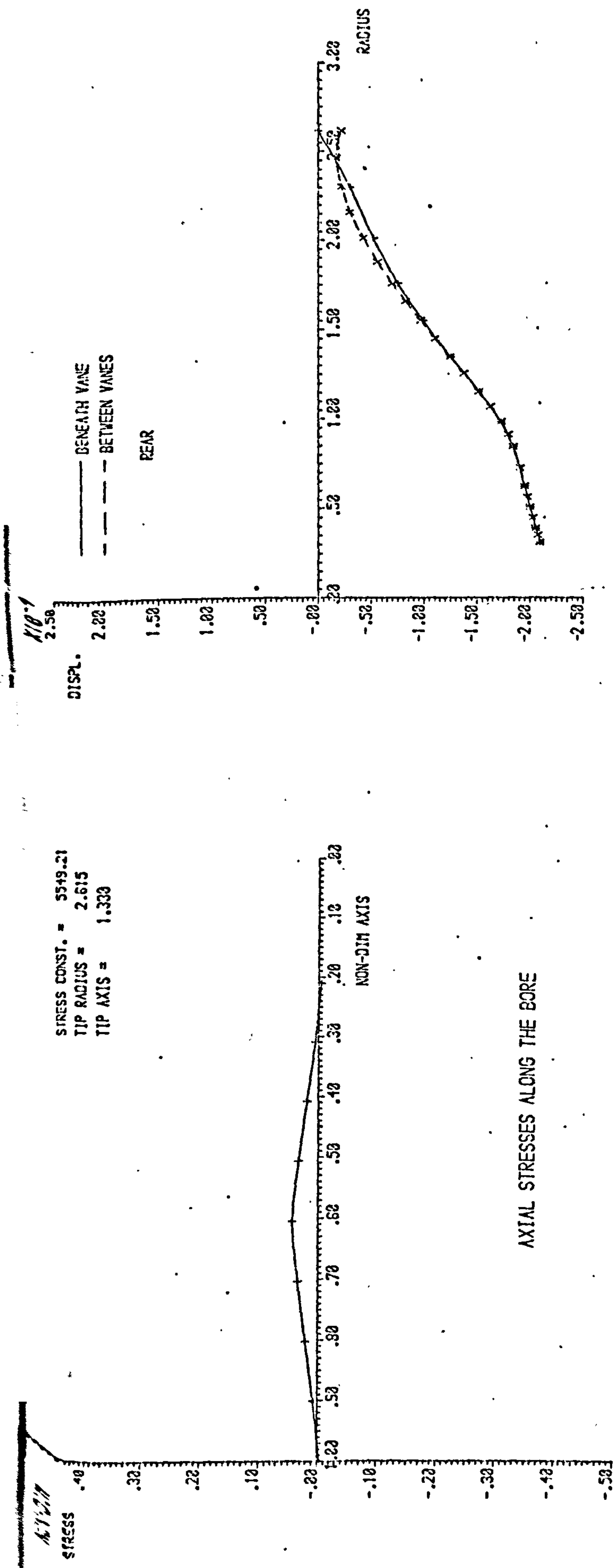
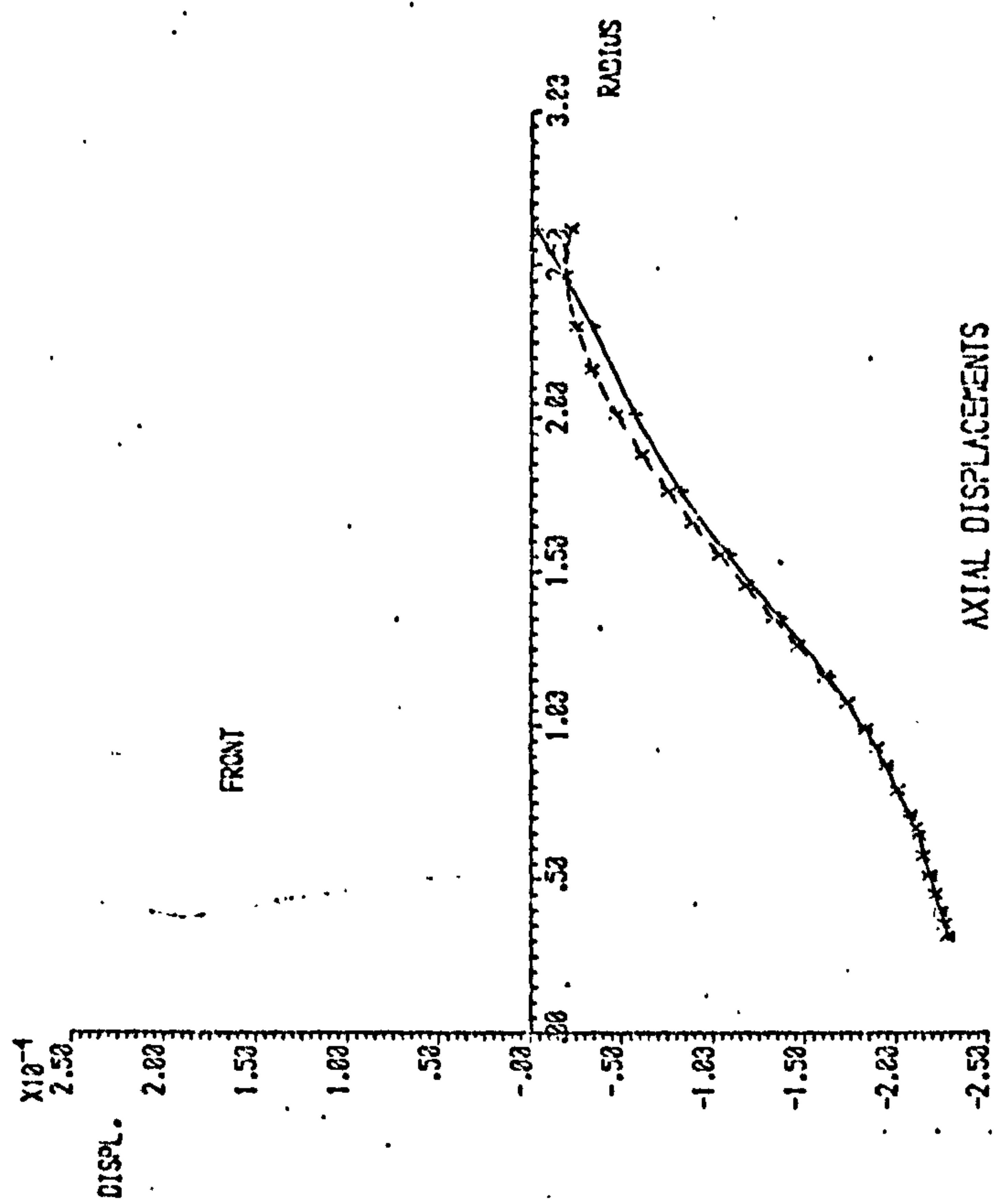
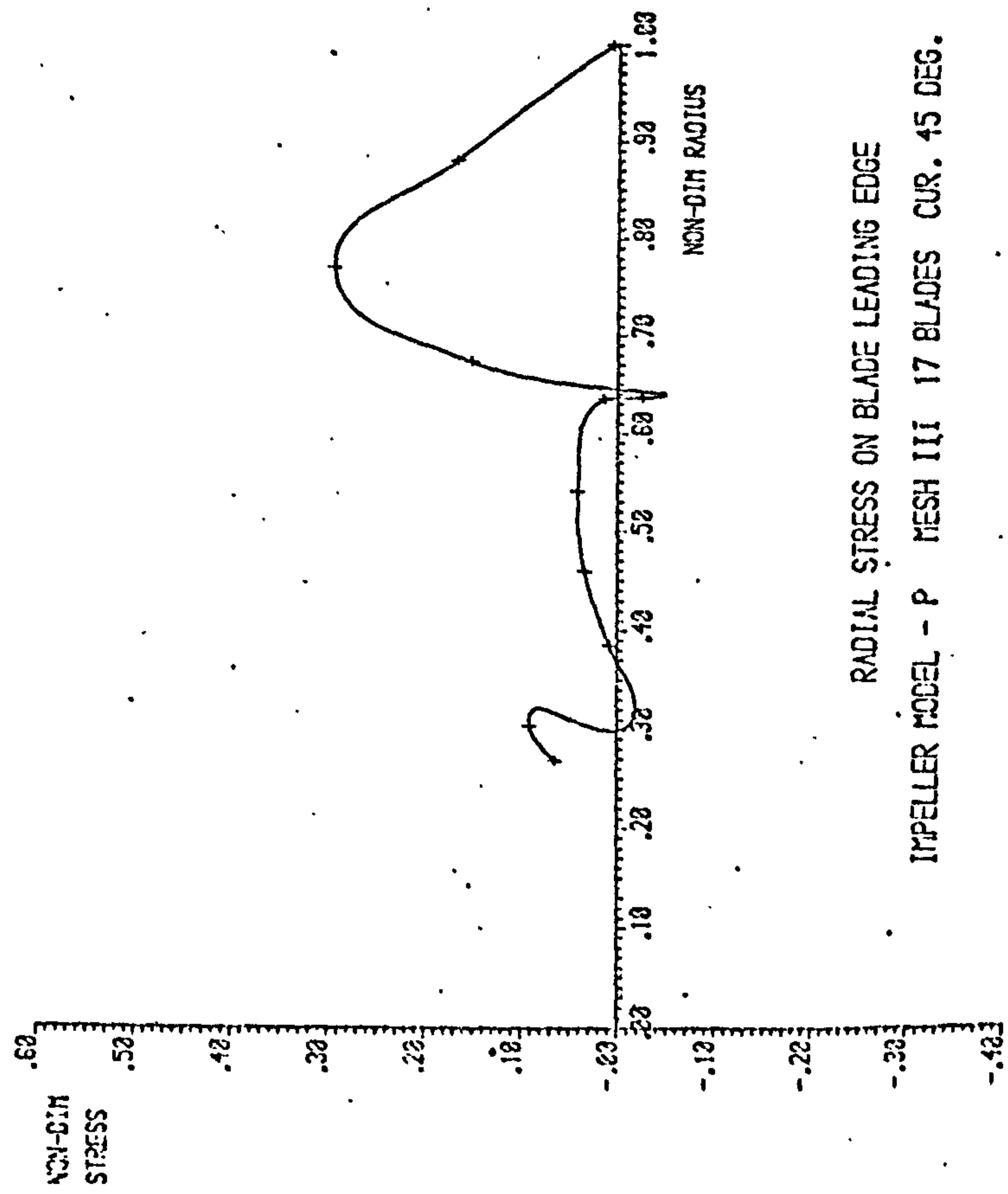
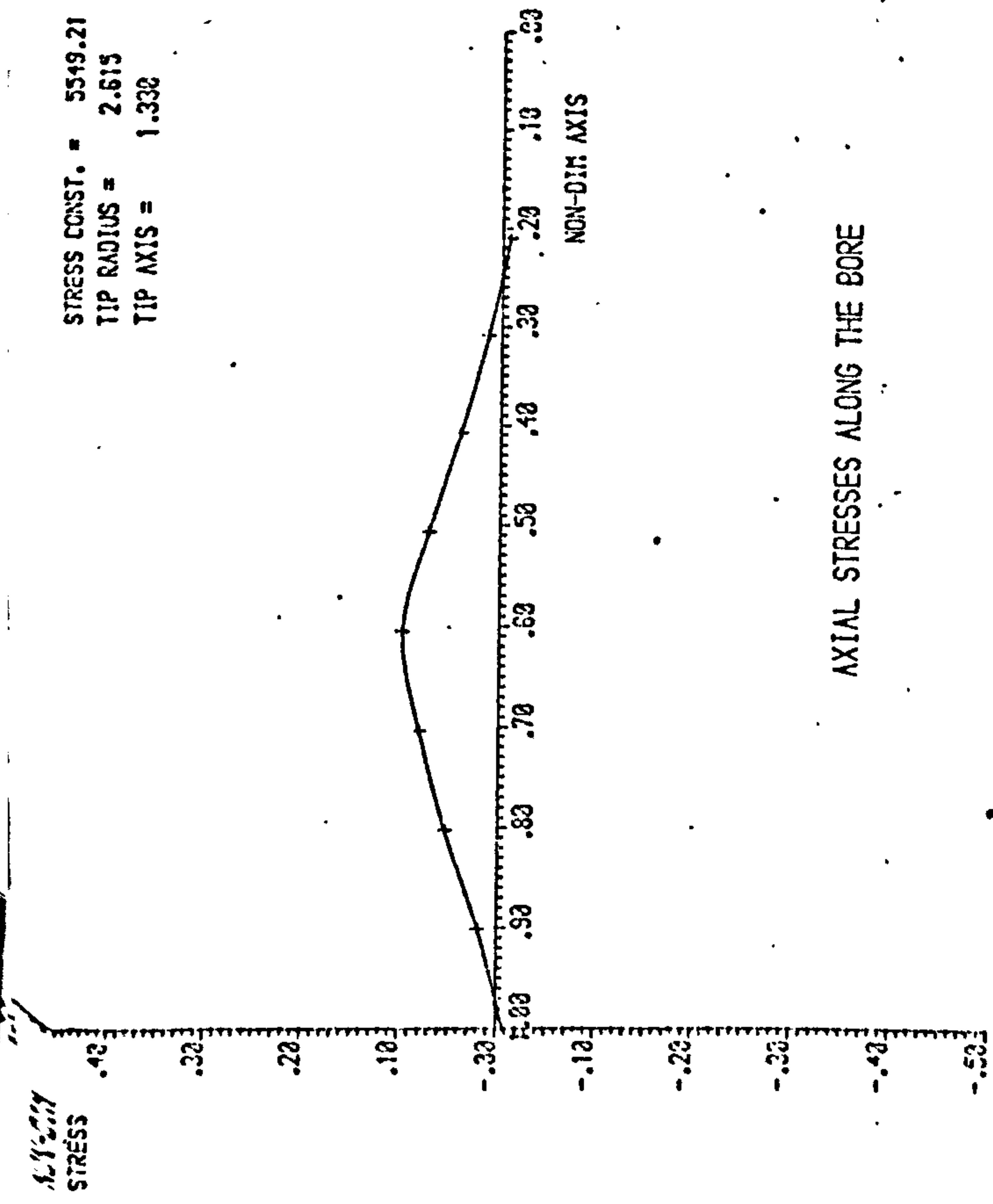
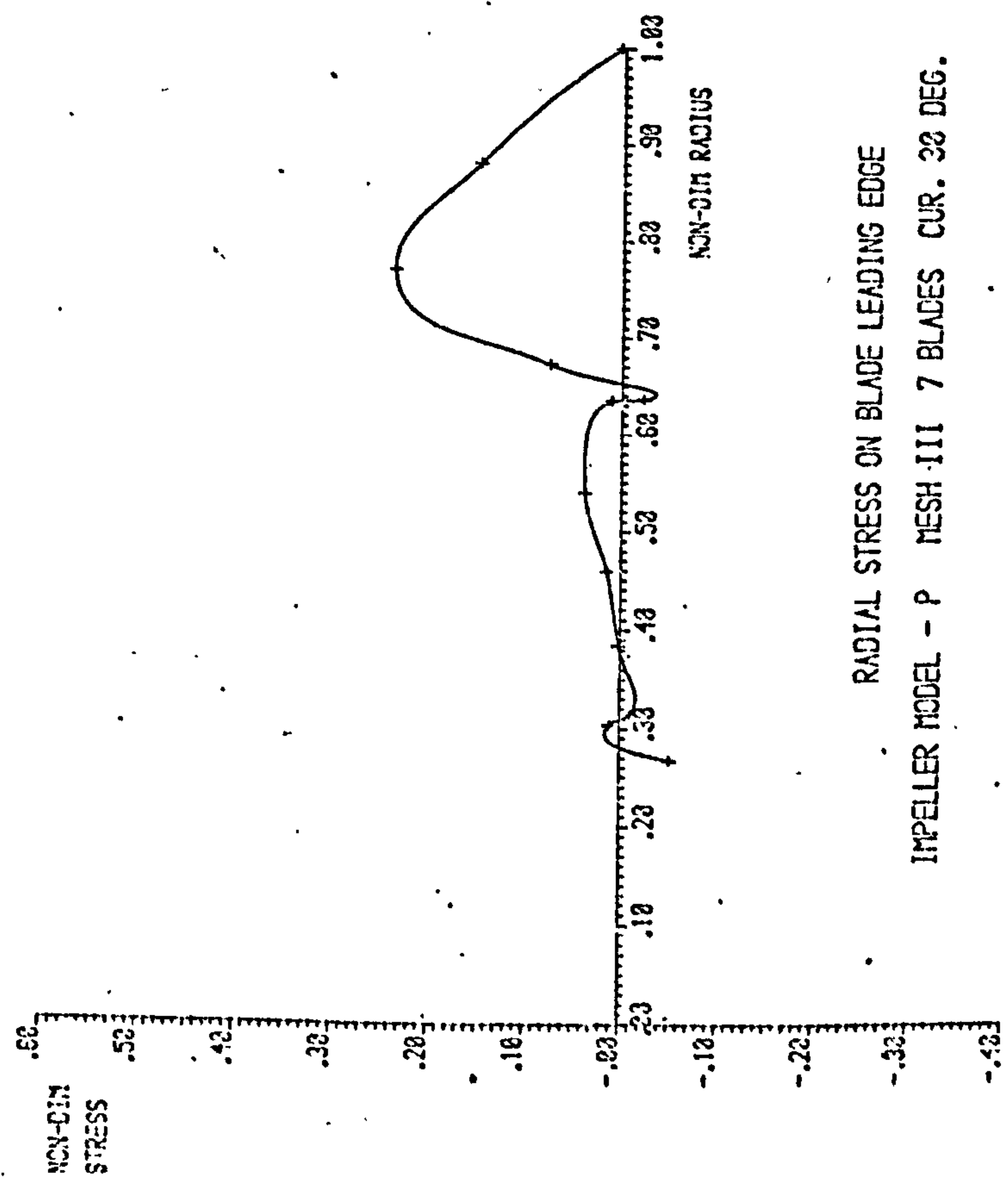
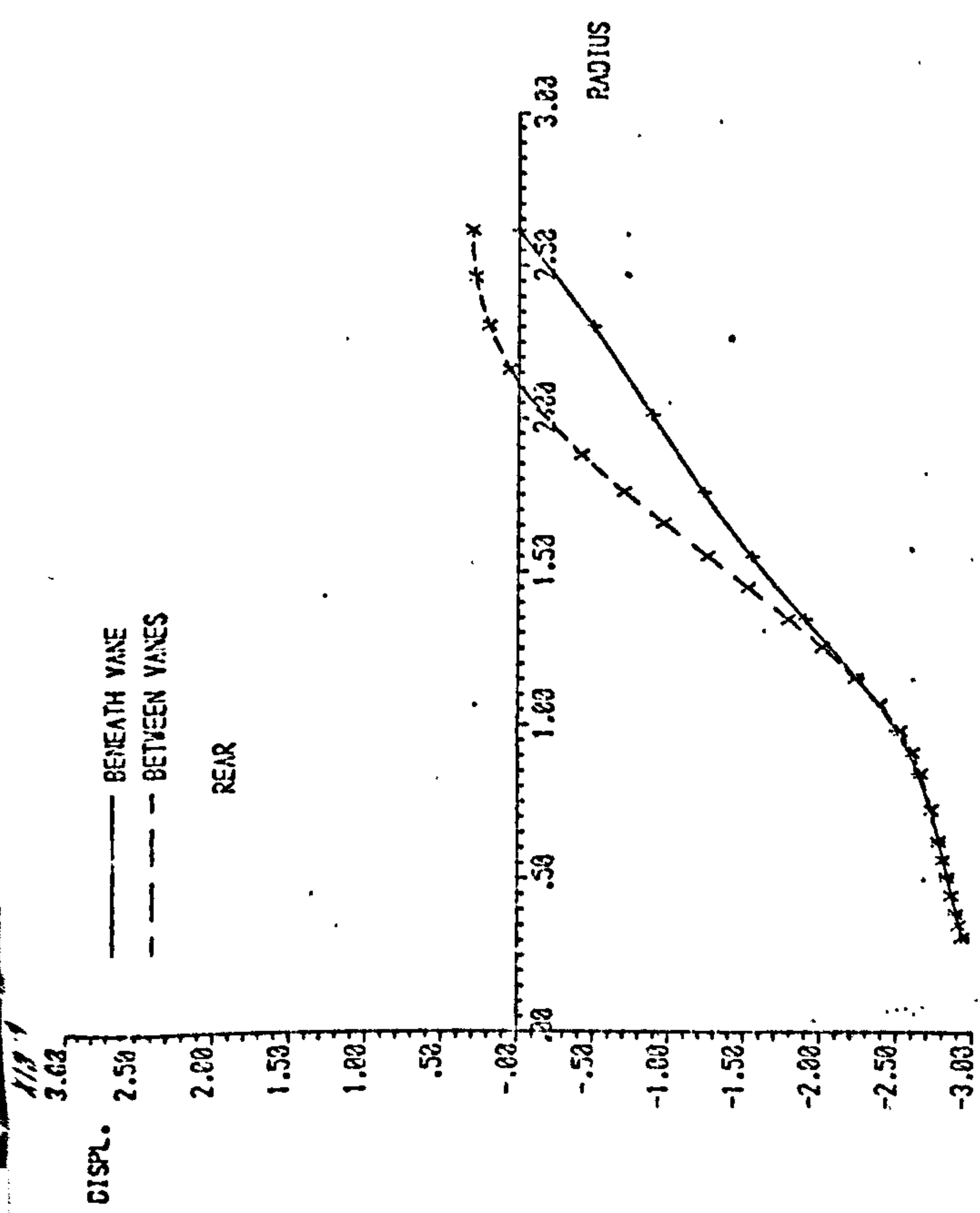
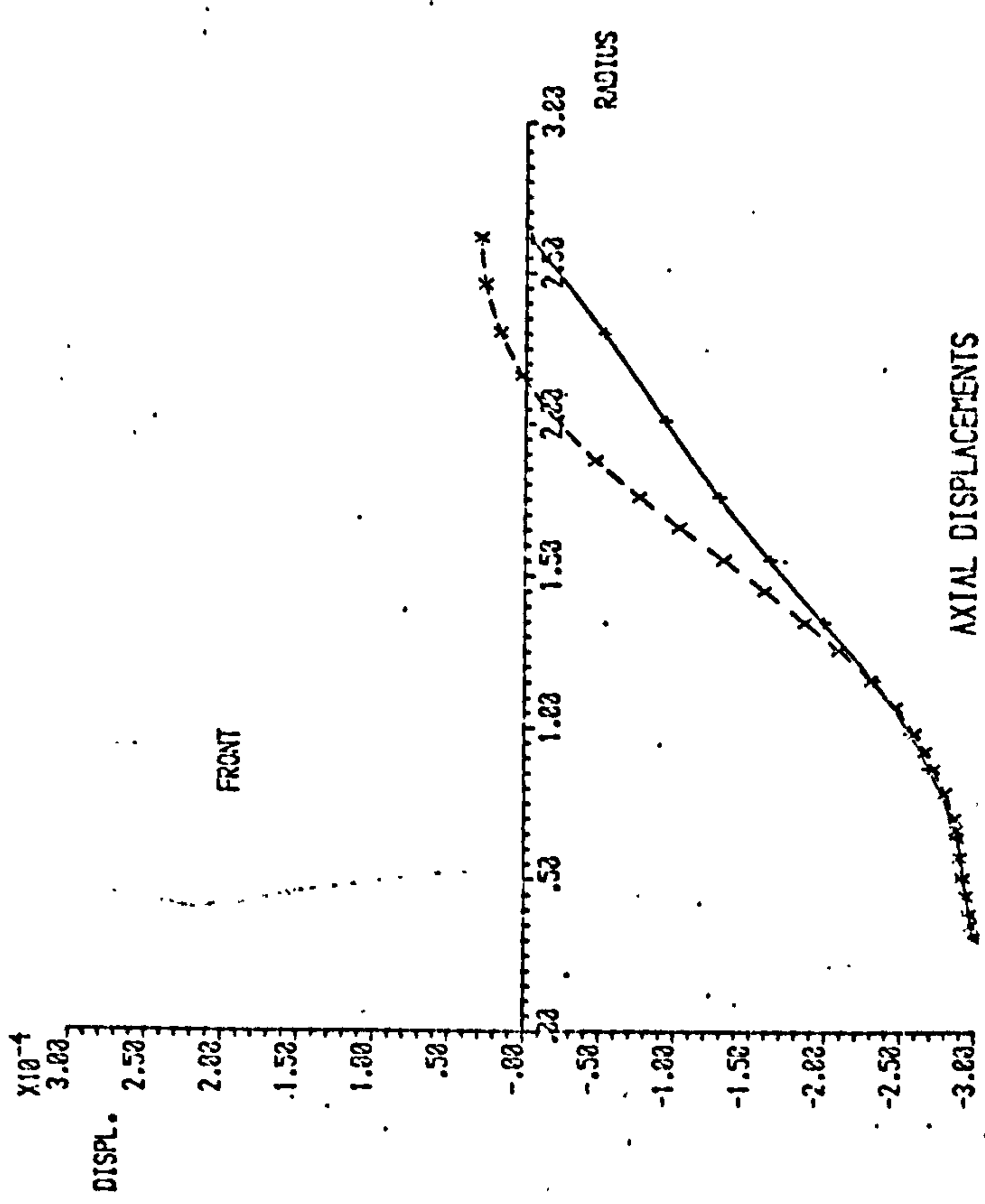


FIG 4-20(f)



AXIAL DISPLACEMENTS
IMPELLER MODEL - P MESH III 7 BLADES CUR. 30 DEG.

FIG 4-20(g)

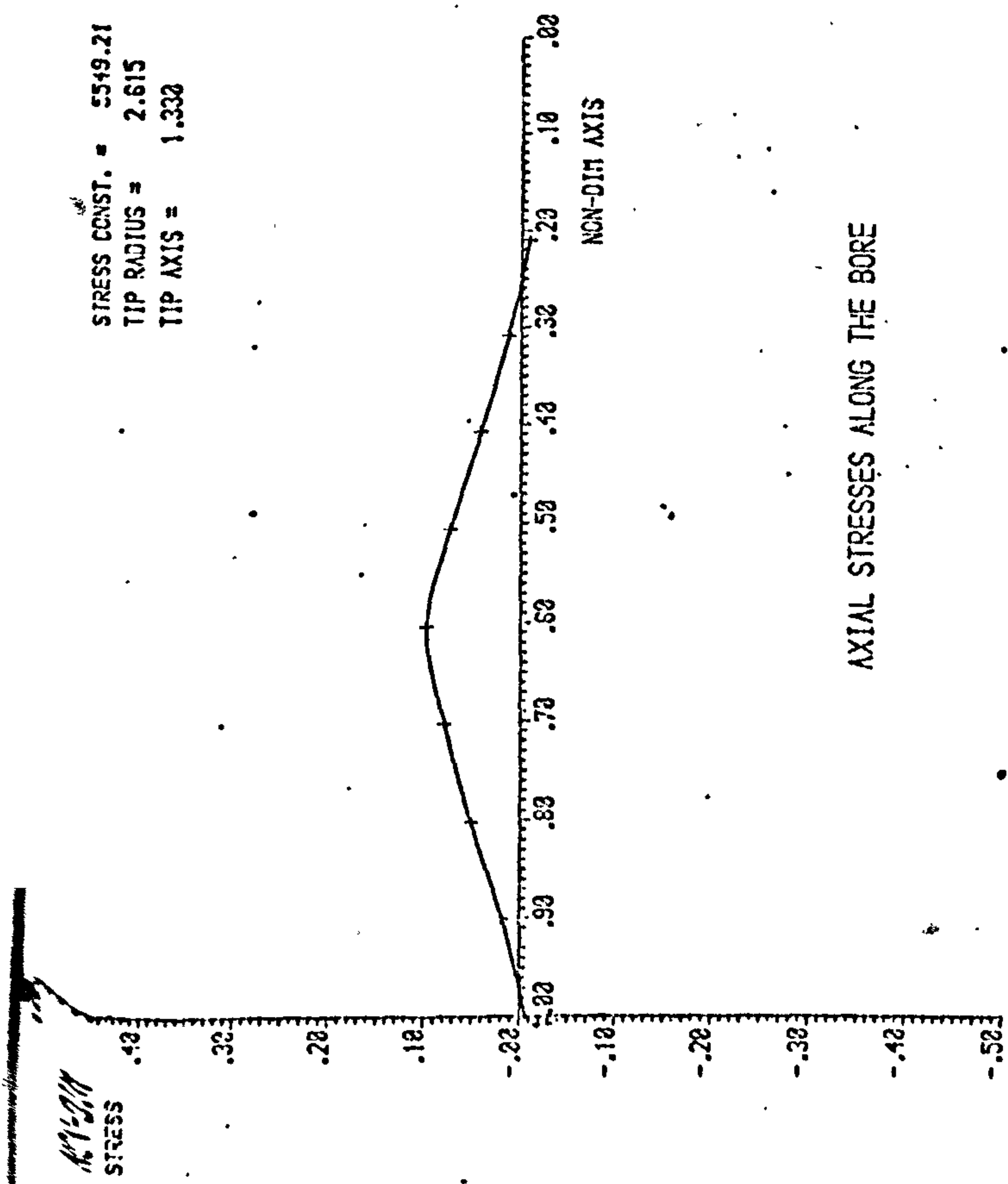
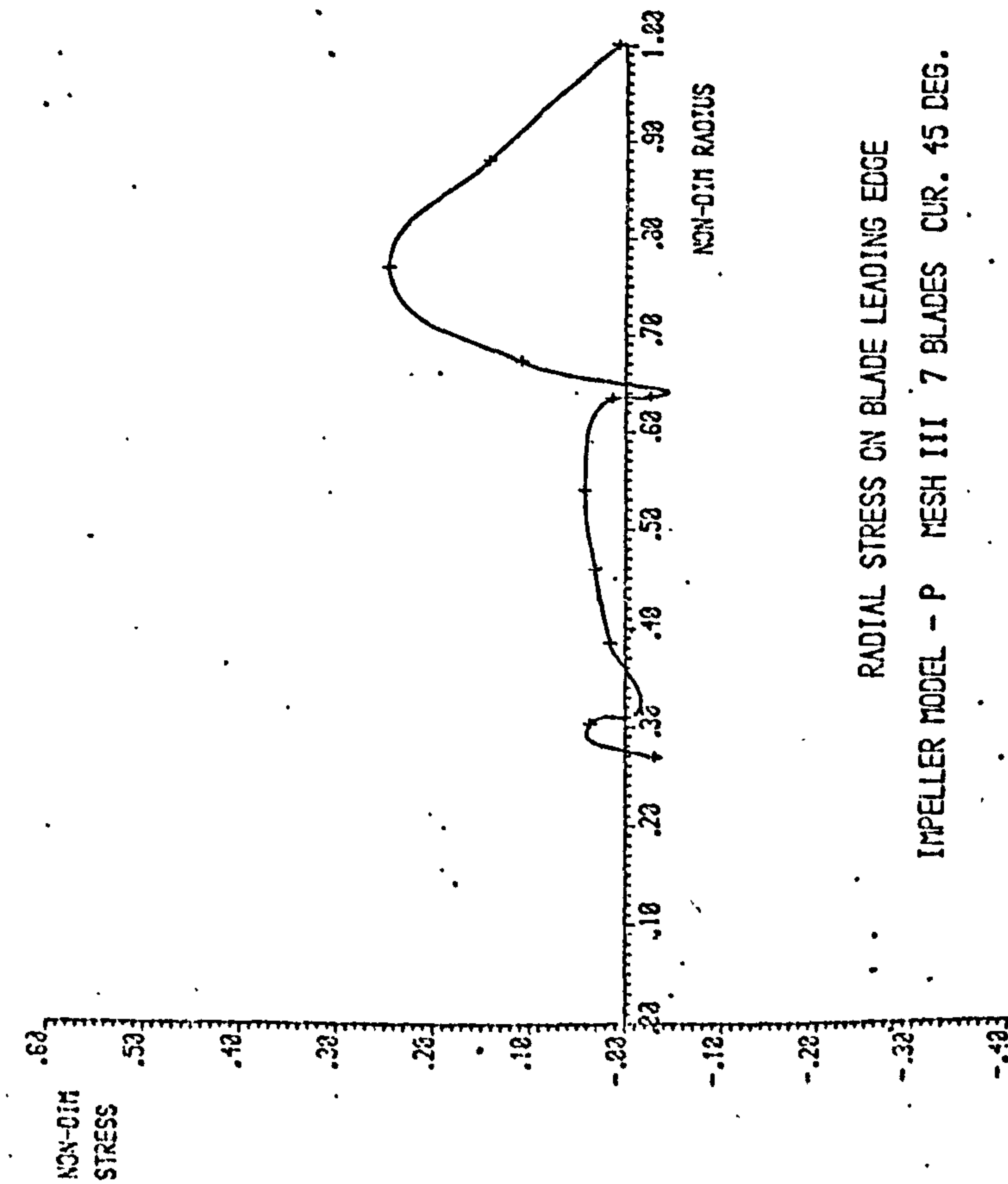
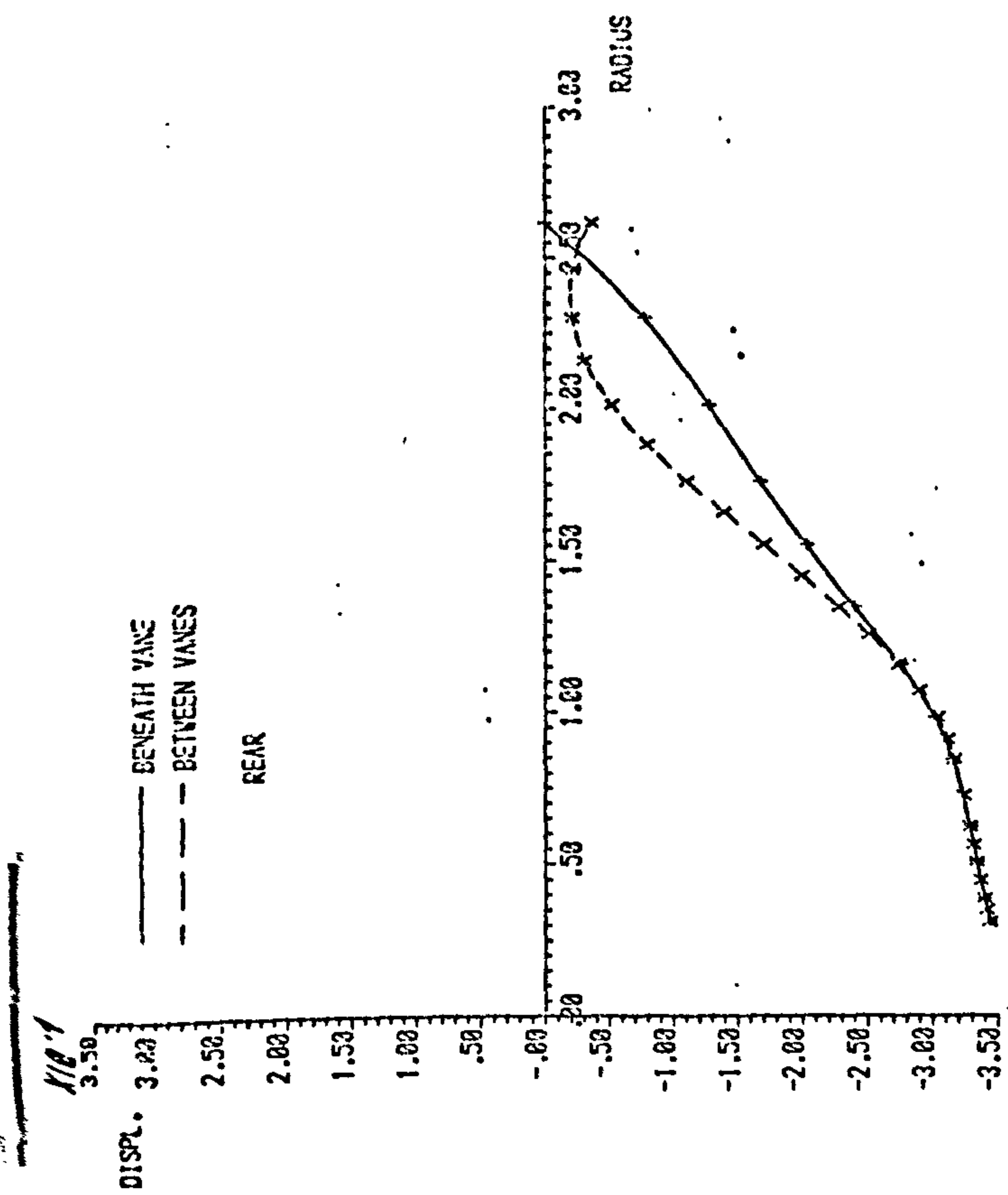
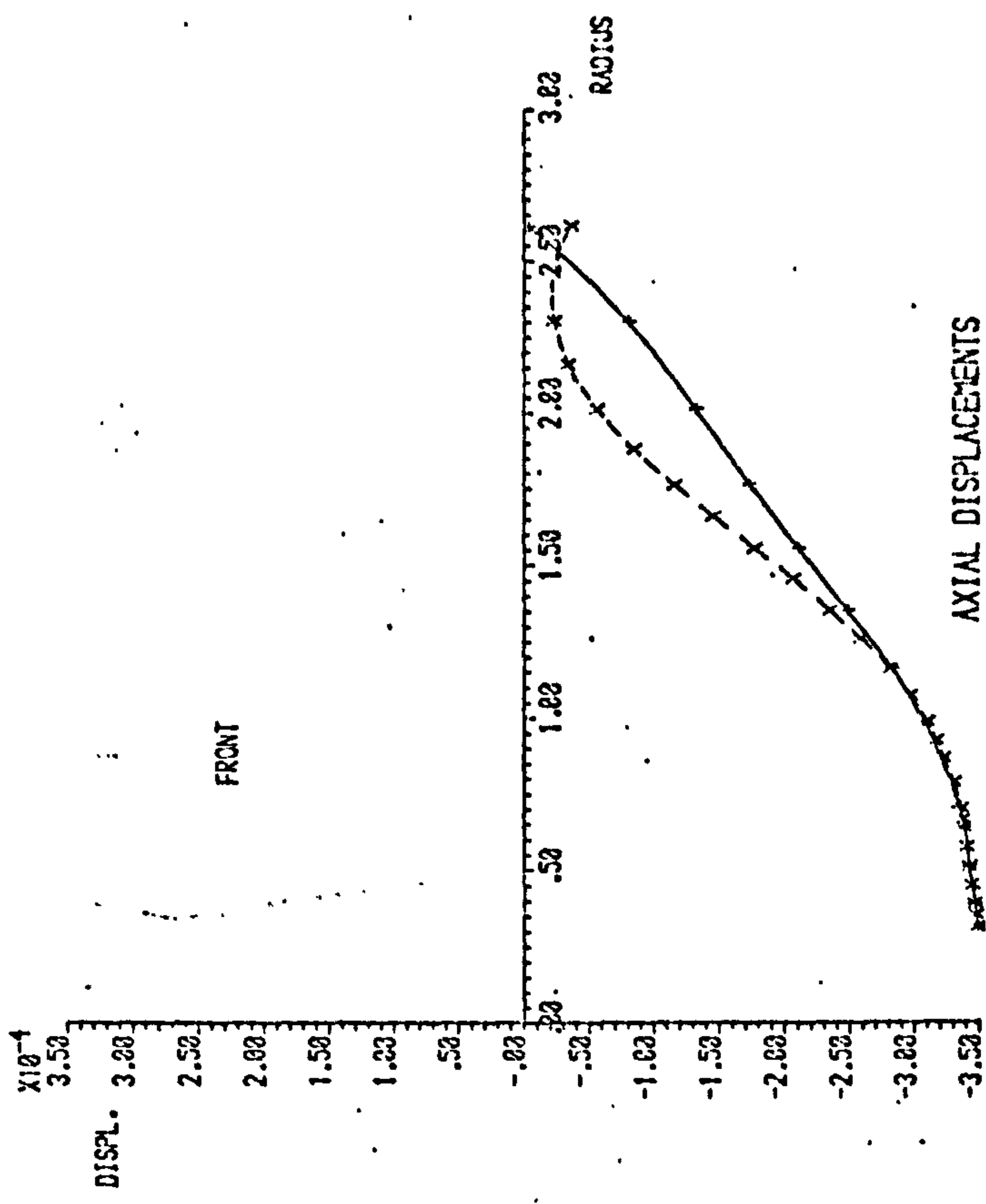
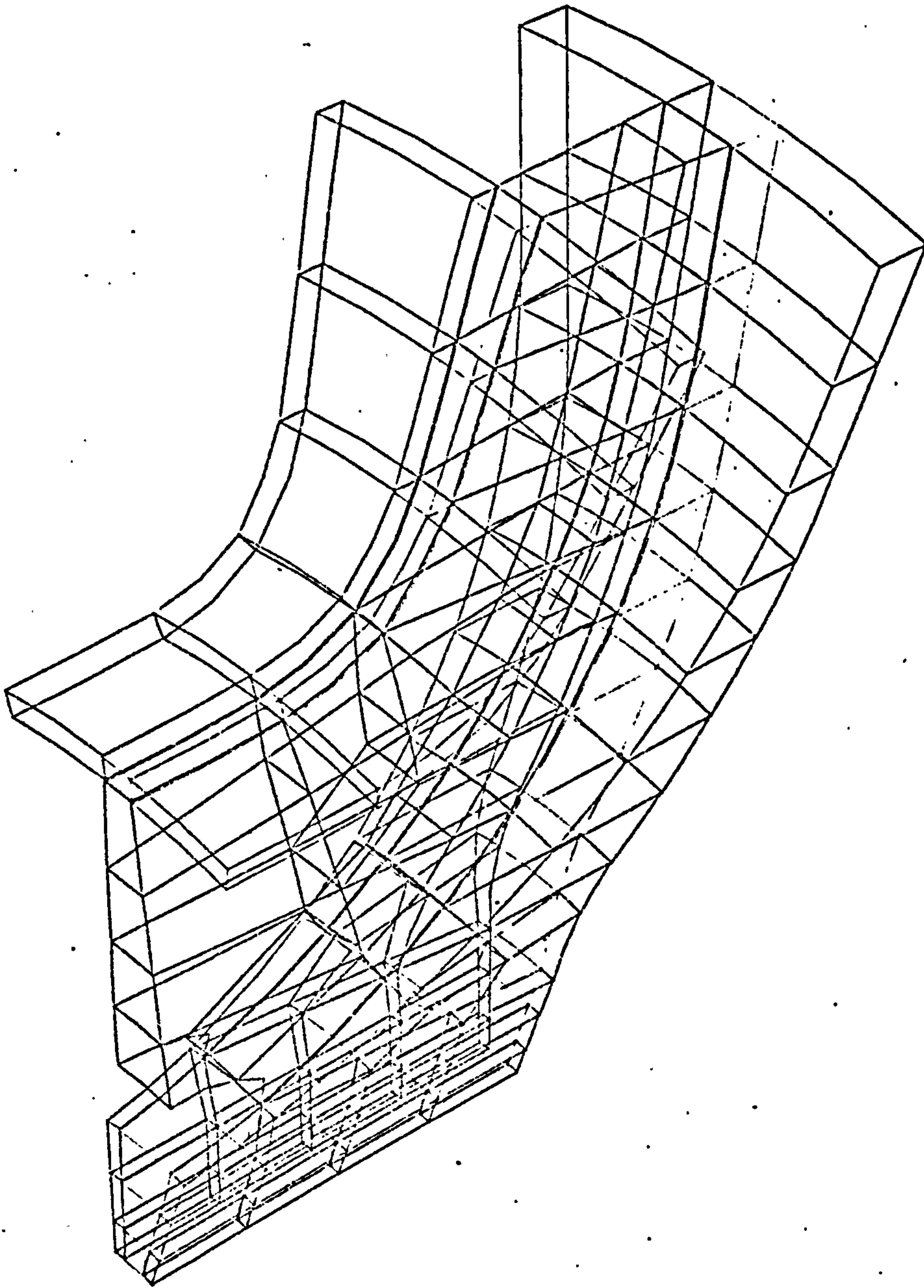


FIG 4-20 (h)

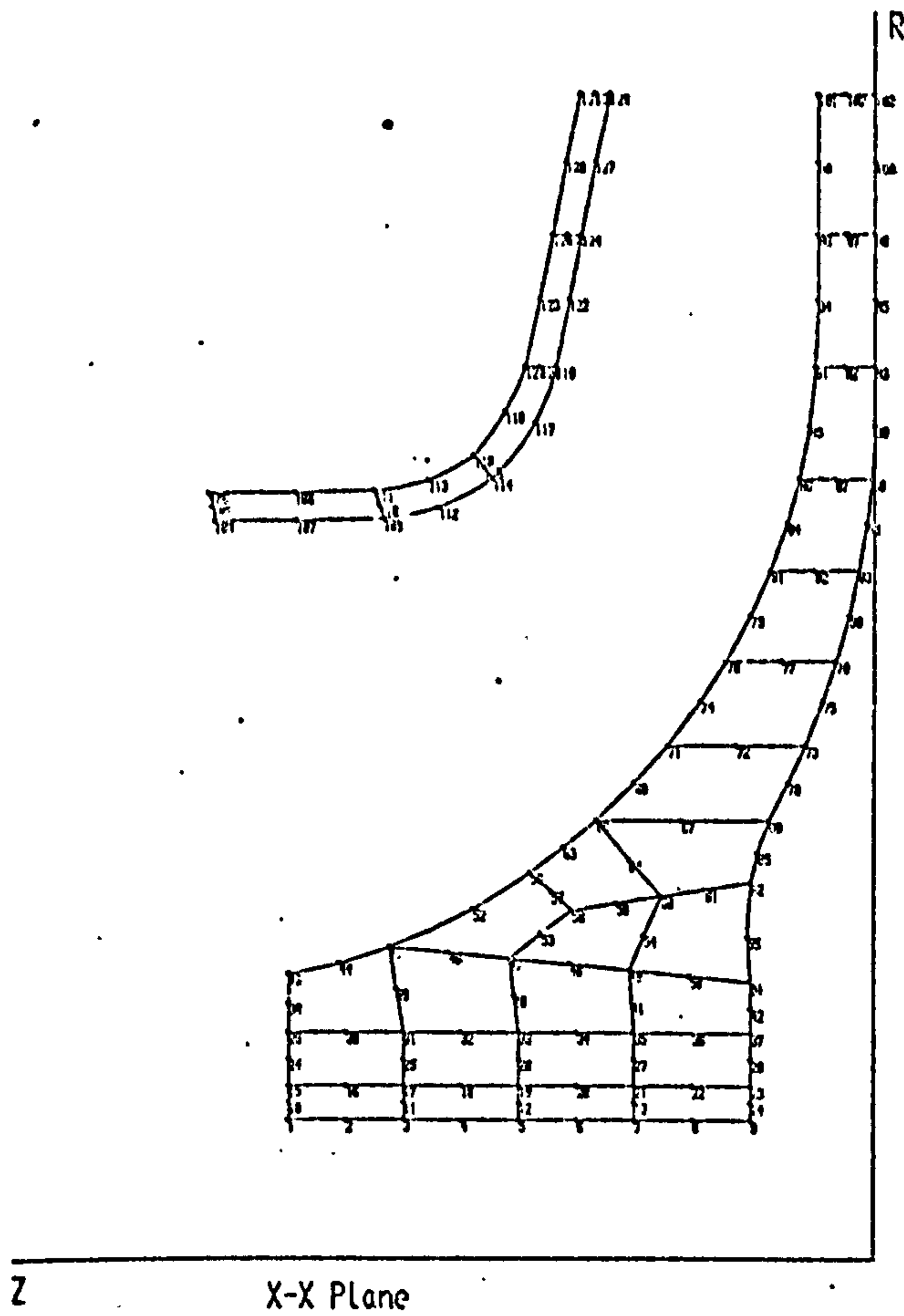
IMPELLER MODEL - P MESH III 7-BLADES CUR. 45 DEG.



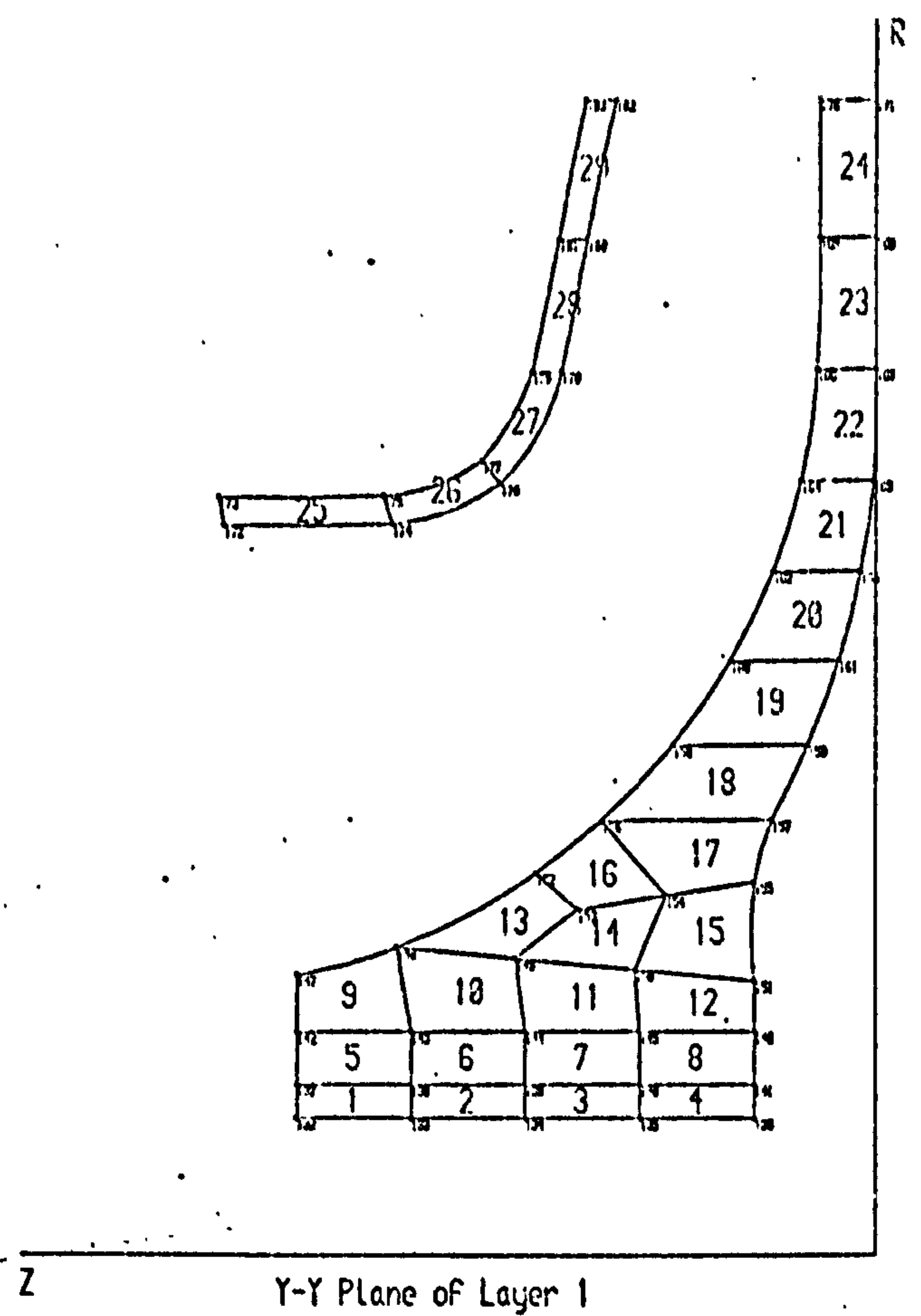
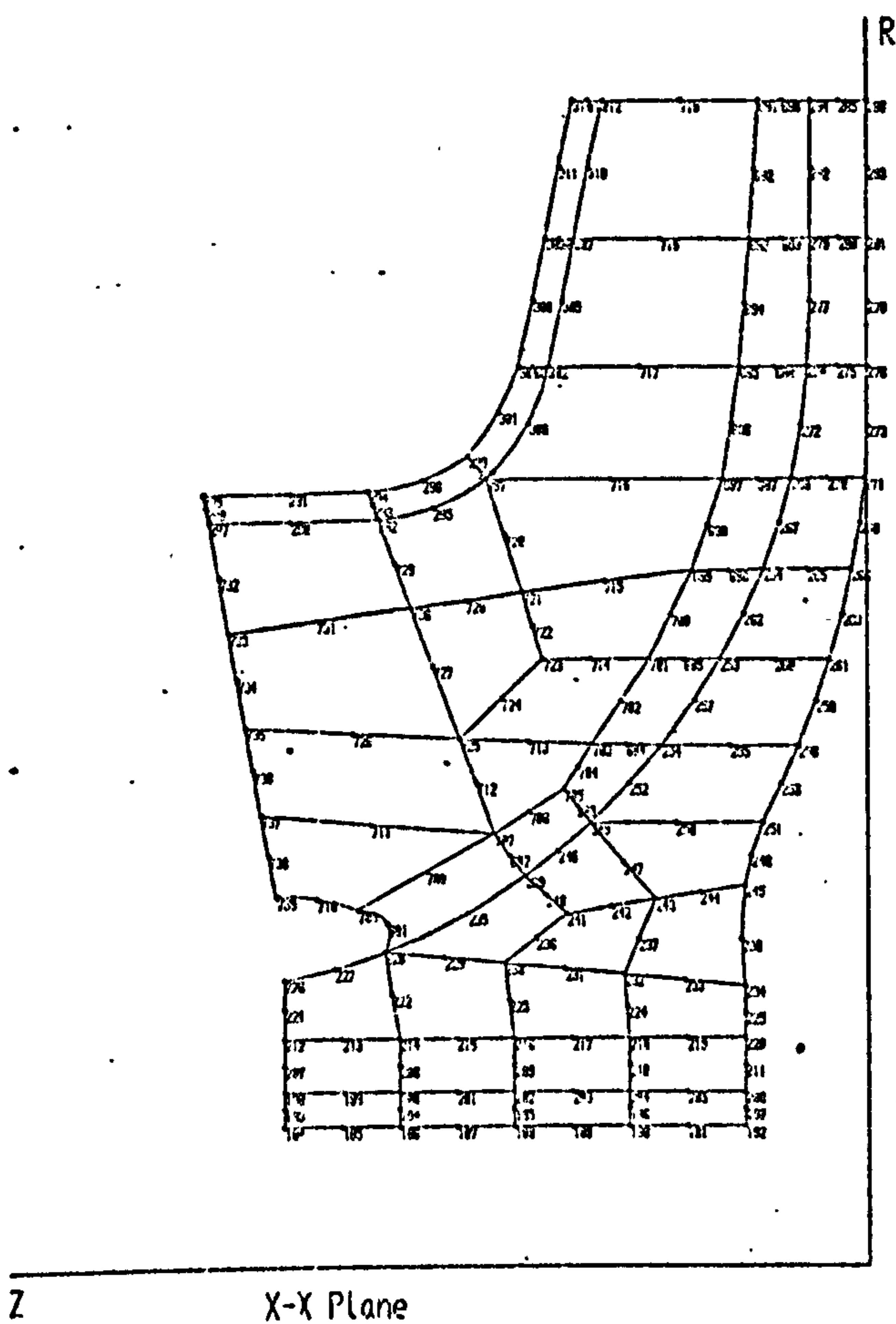
3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-P WITH COVER DISK MESH III

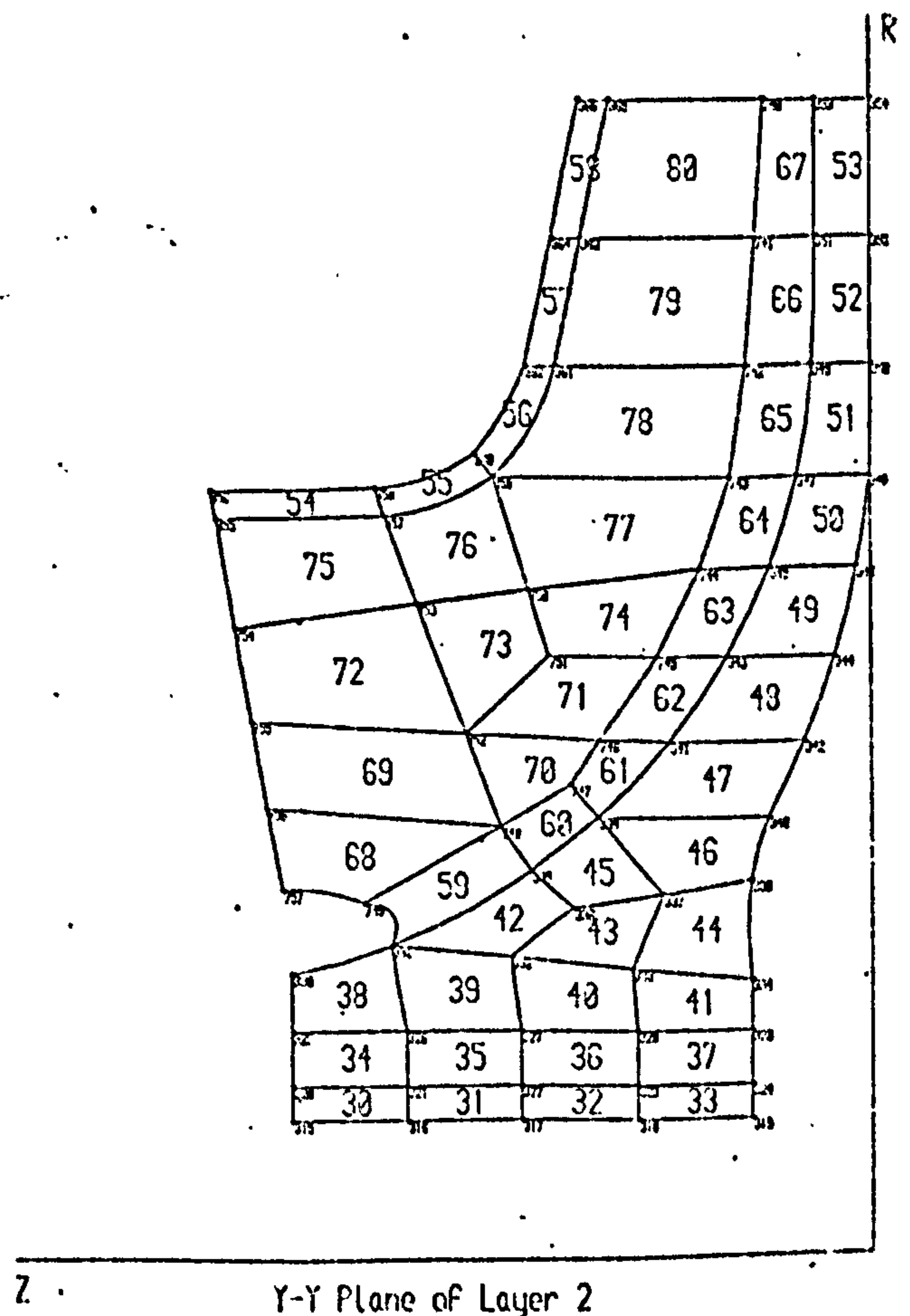
FIG 4-21(a)



IMPELLER MODEL-P WITH COVER DISK MESH III

DRAWN IN MM SCALE: 55.03 TIMES THE SUPPLIED UNITS
Sheet 1

IMPELLER MODEL-P WITH COVER DISK MESH III

DRAWN IN MM SCALE: 55.03 TIMES THE SUPPLIED UNITS
Sheet 2



X-X Plane

IMPELLER MODEL-P WITH COVER DISK MESH III



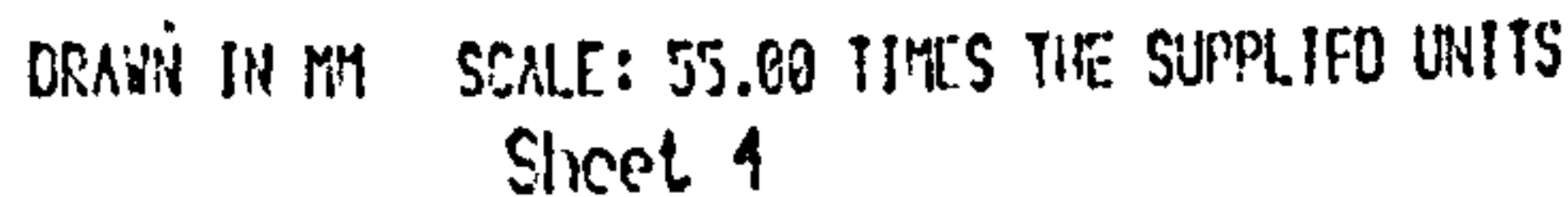
Y-Y Plane of Layer 3

DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS
Sheet 3

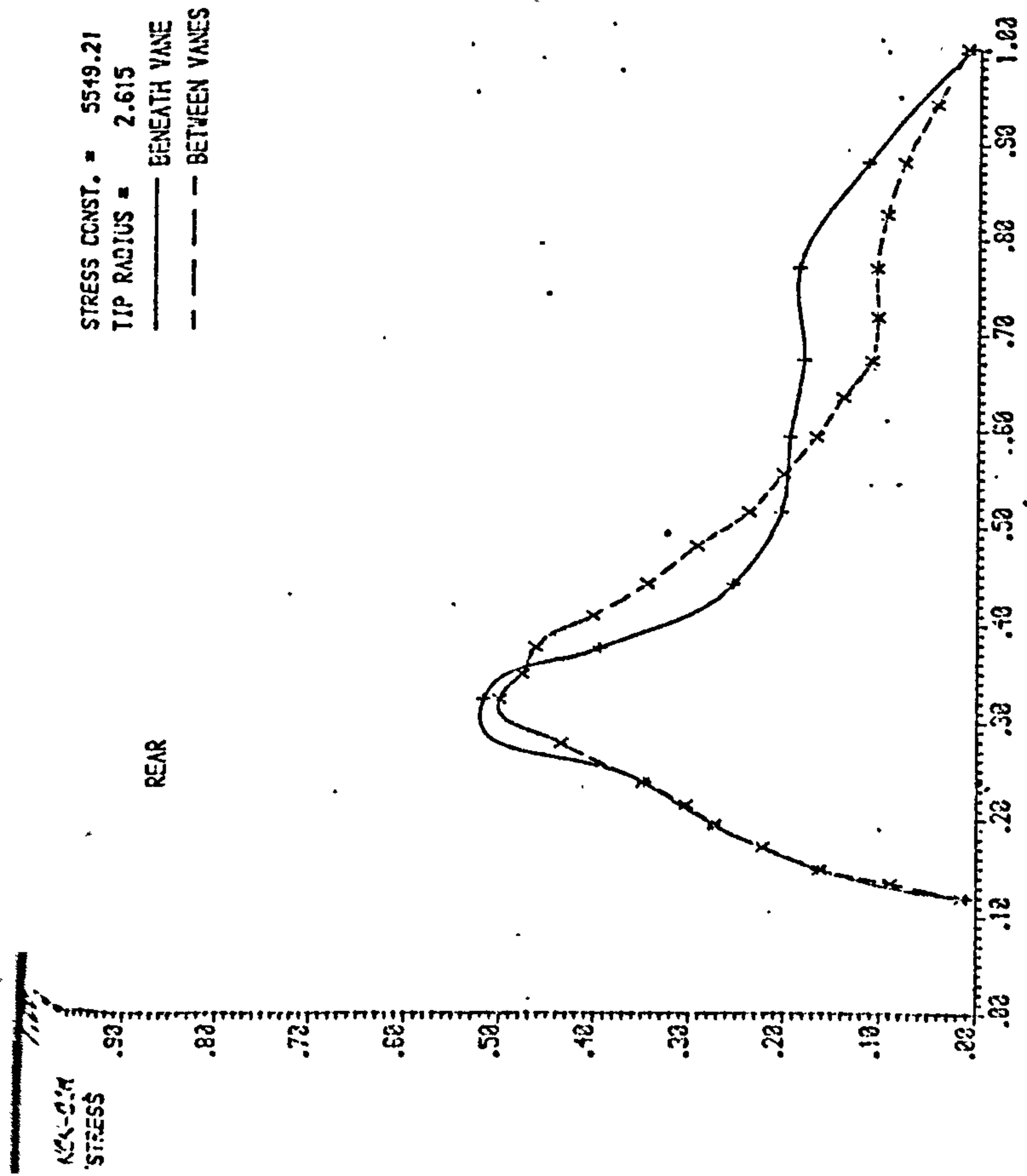
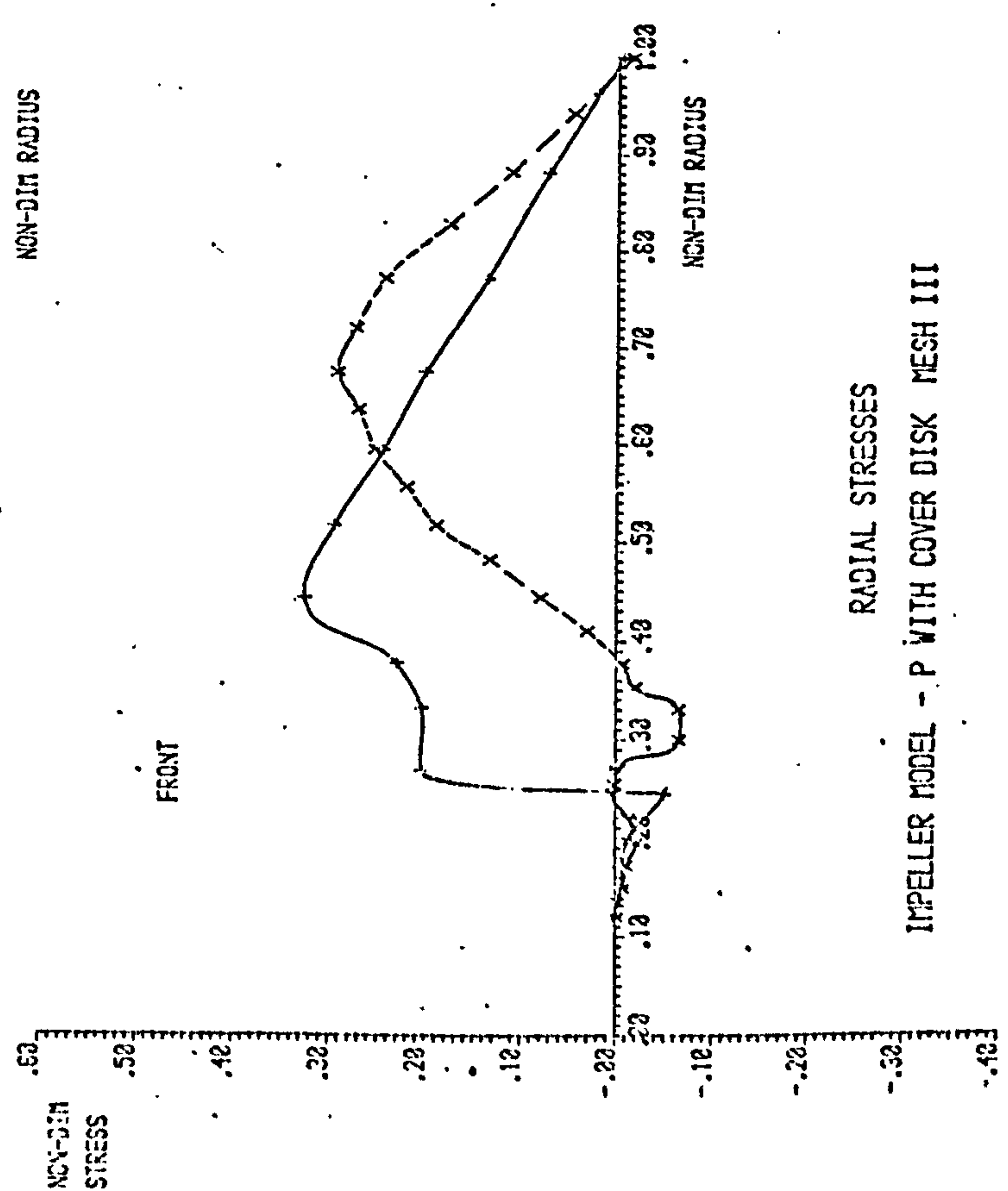


X-X Plane

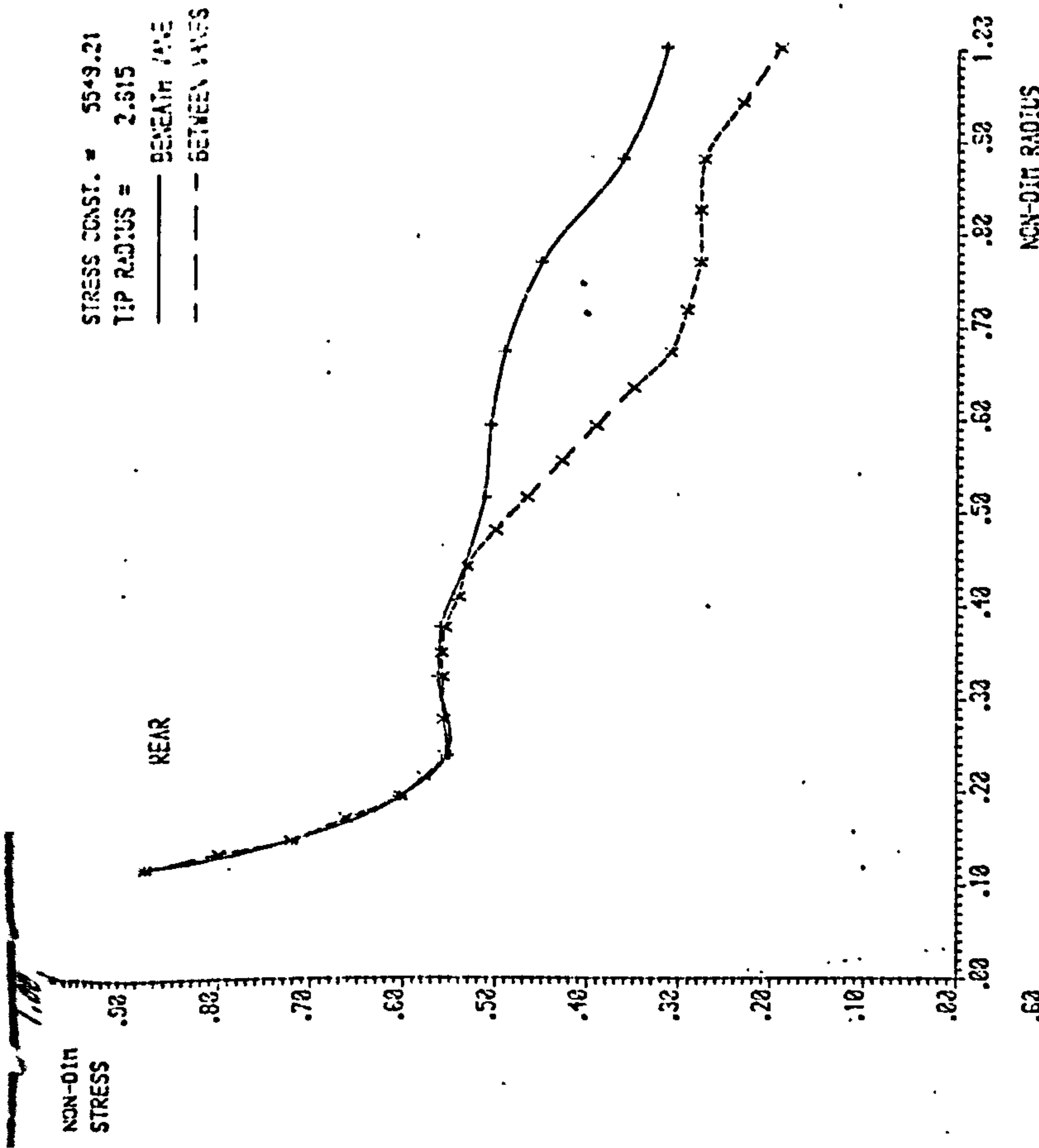
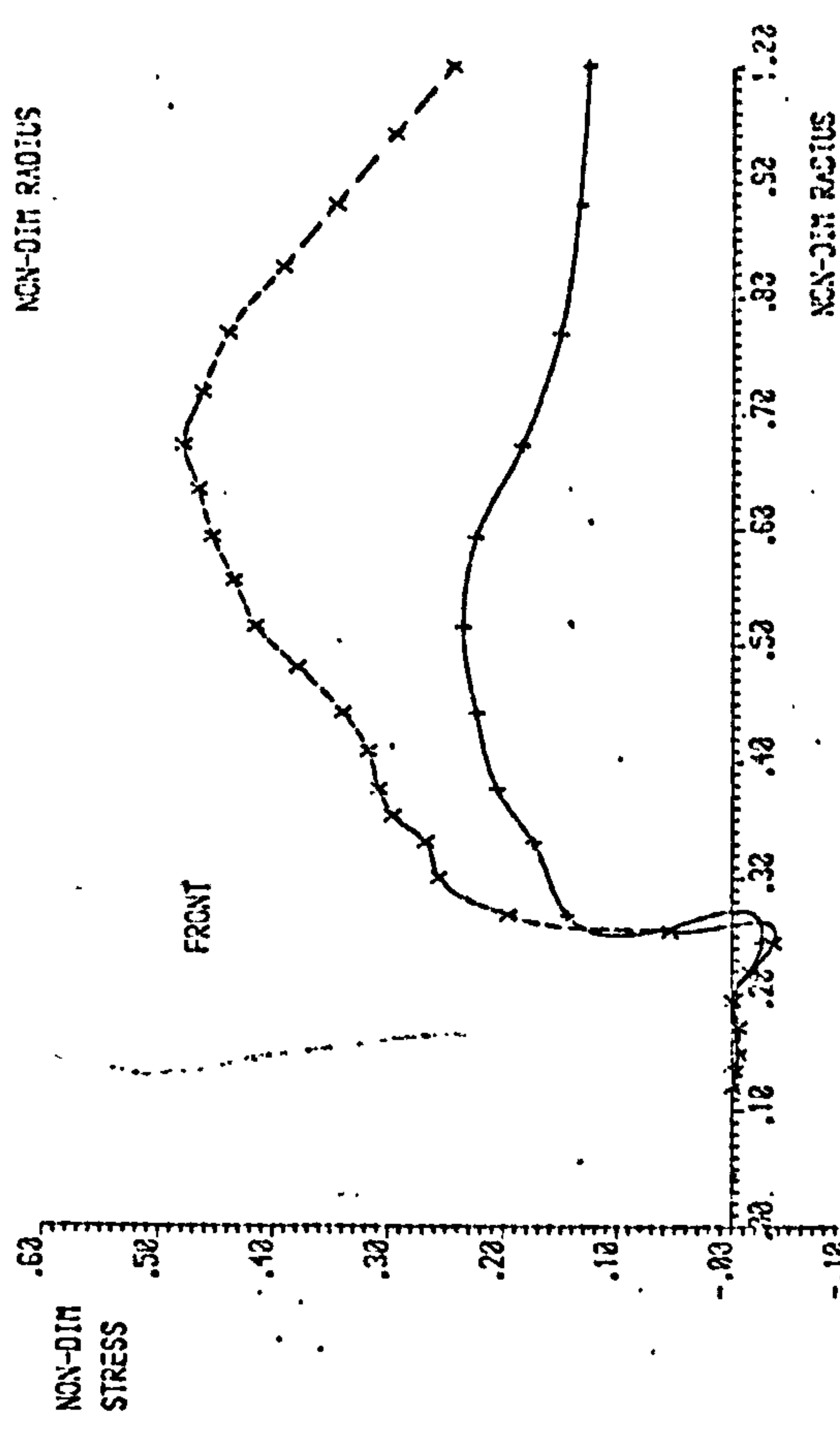
IMPELLER MODEL-P WITH COVER DISK MESH III



TEXT BOUND INTO THE SPINE

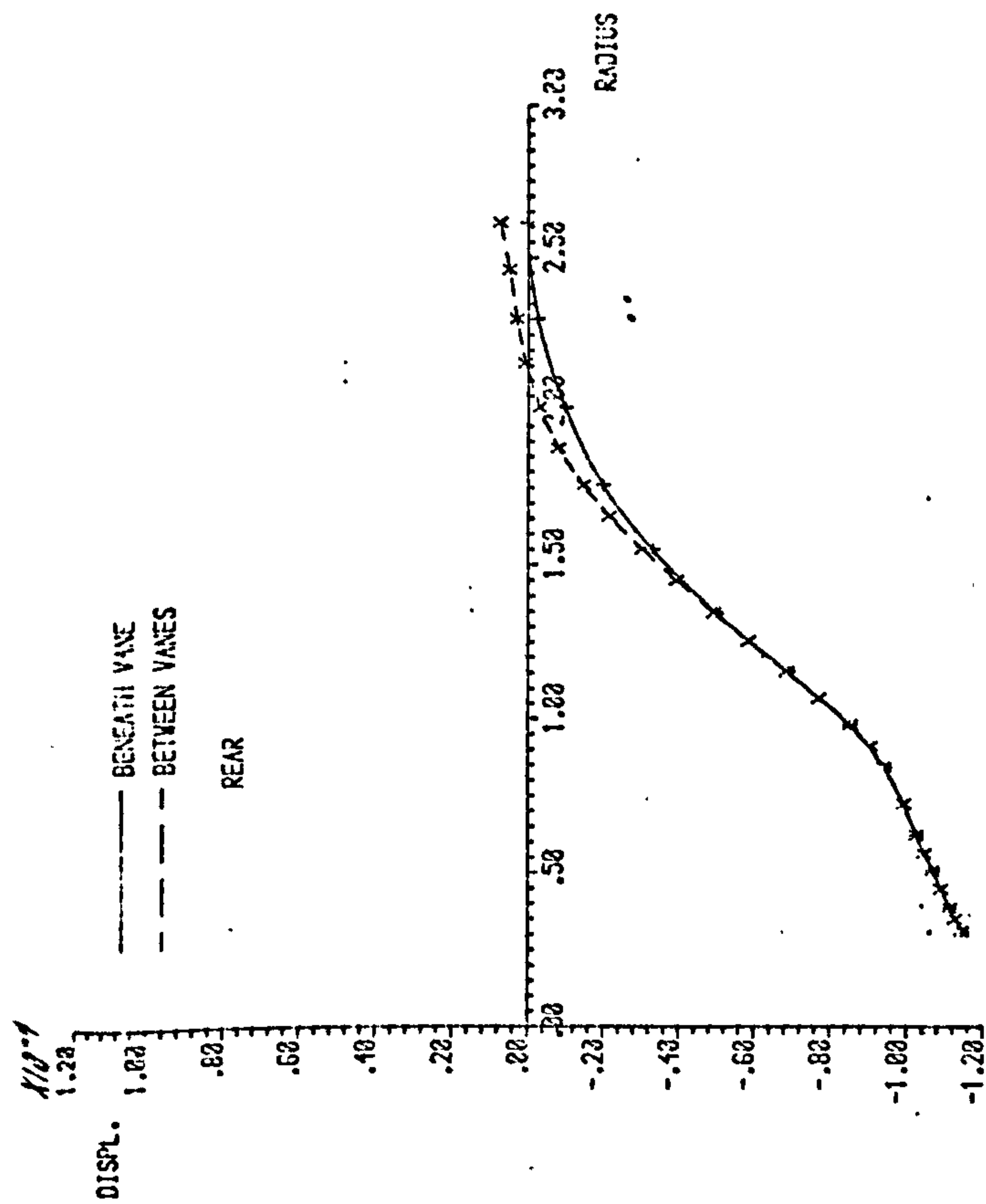
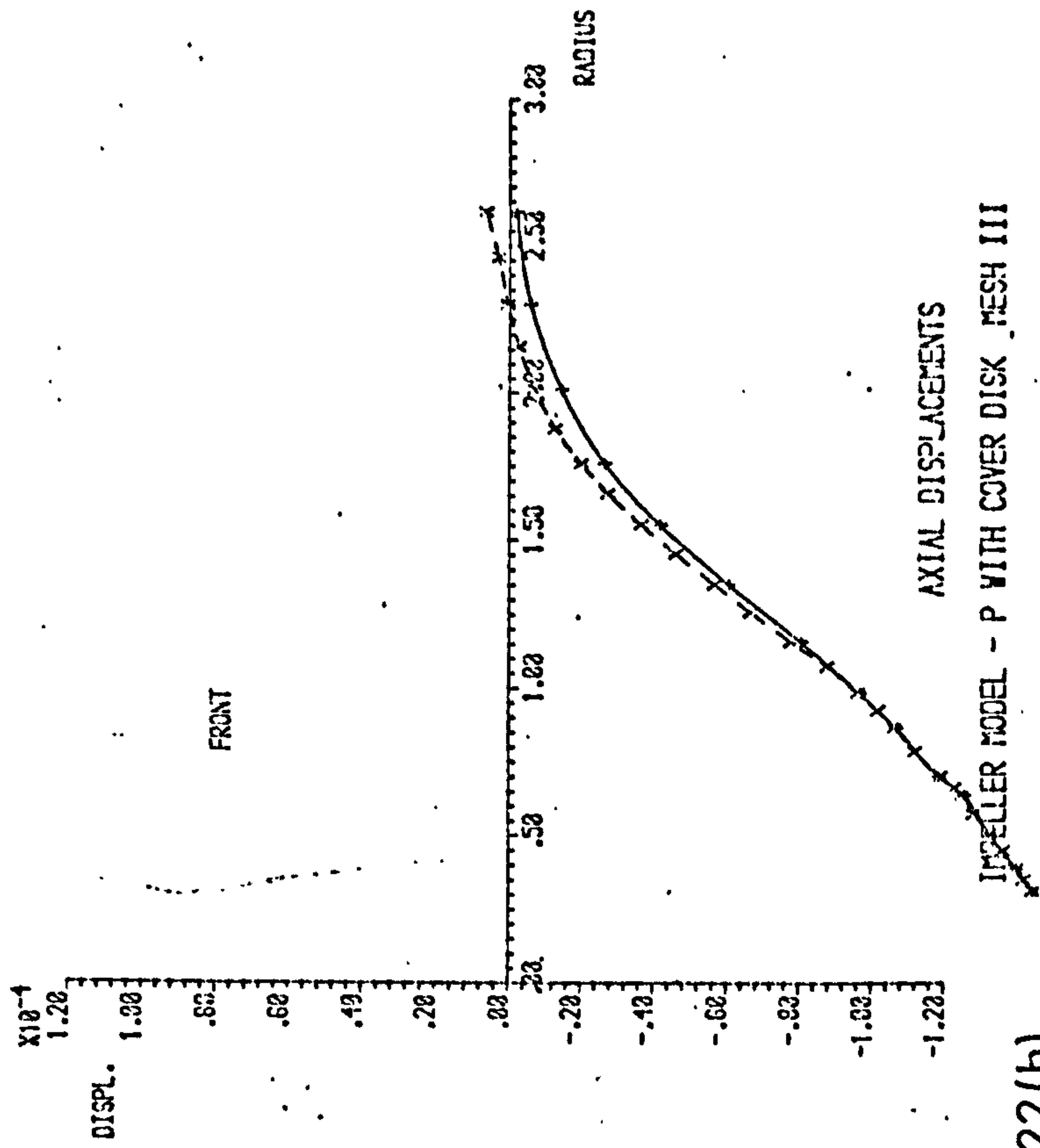


RADIAL STRESSES
IMPELLER MODEL - P WITH COVER DISK MESH III



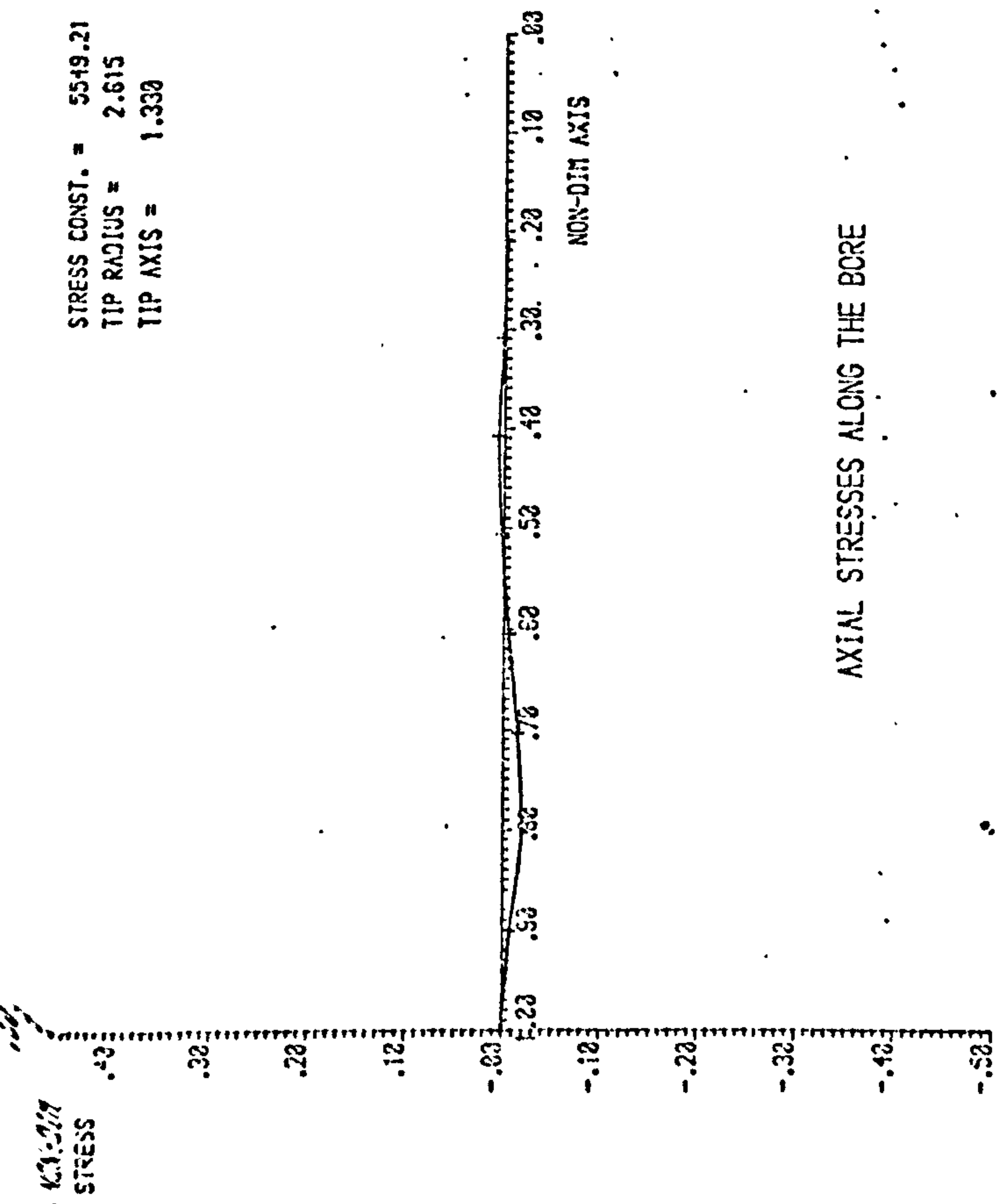
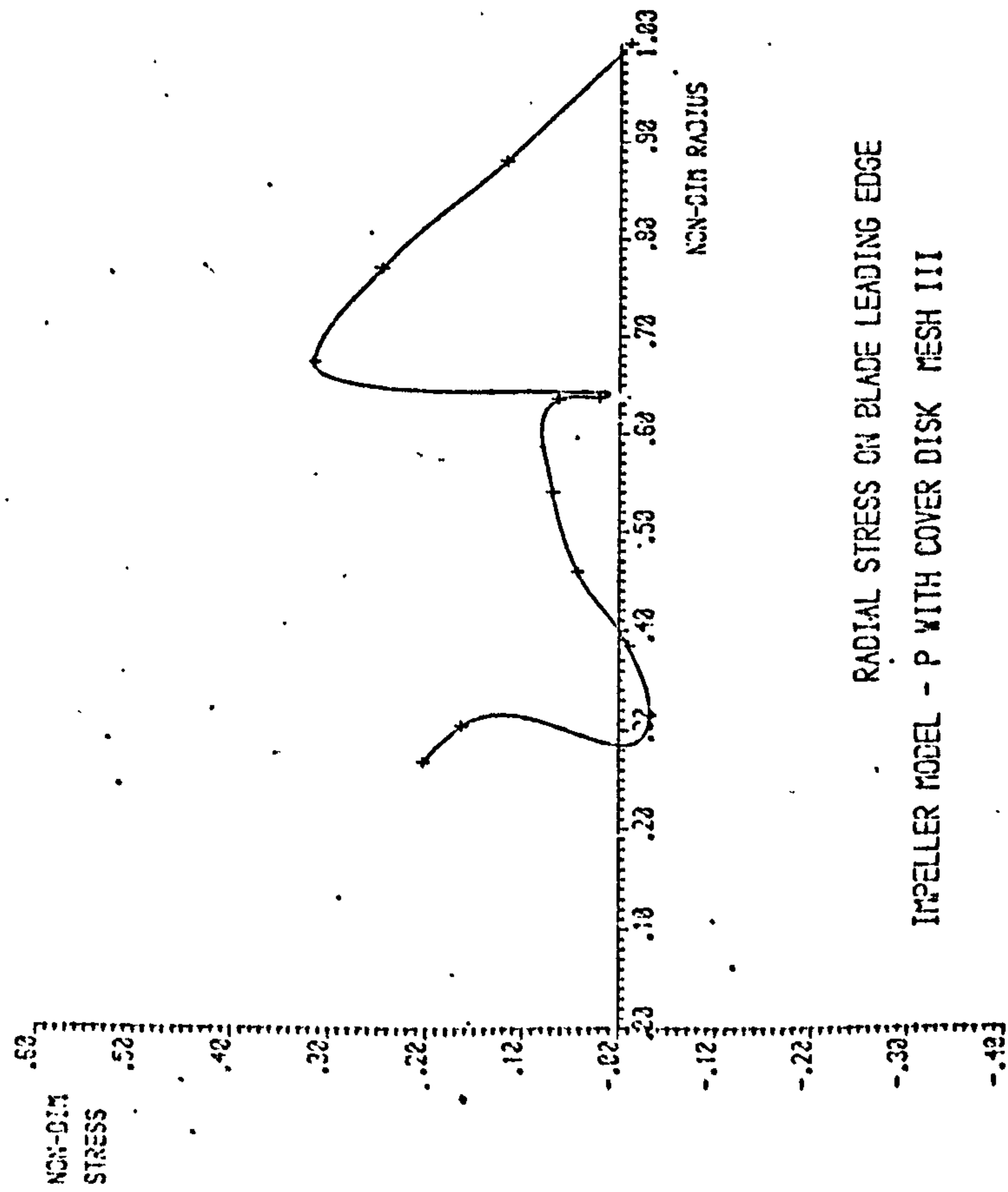
CIRCUMFERENTIAL STRESSES
IMPELLER MODEL - P WITH COVER DISK MESH III

FIG 4-22(a)



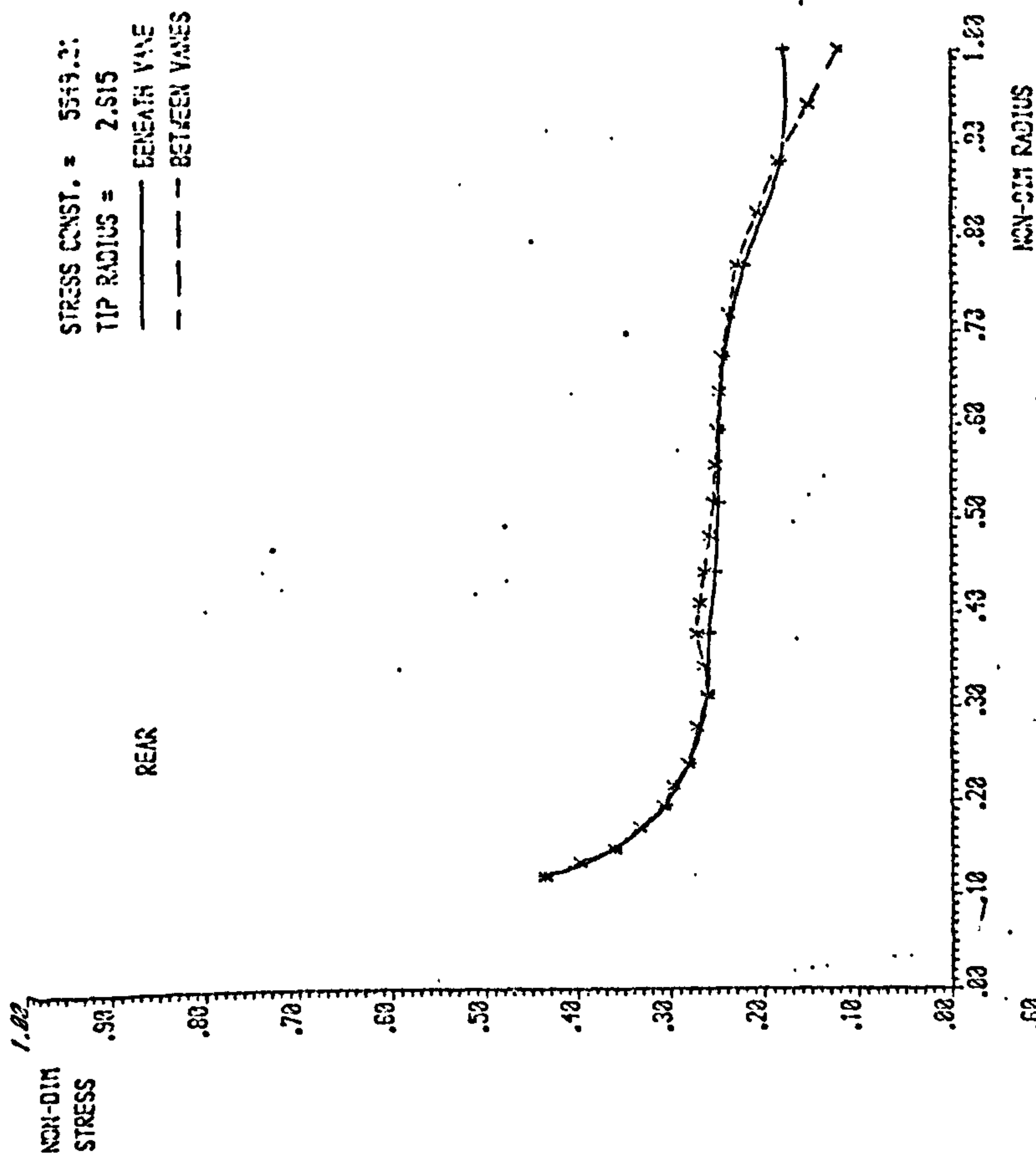
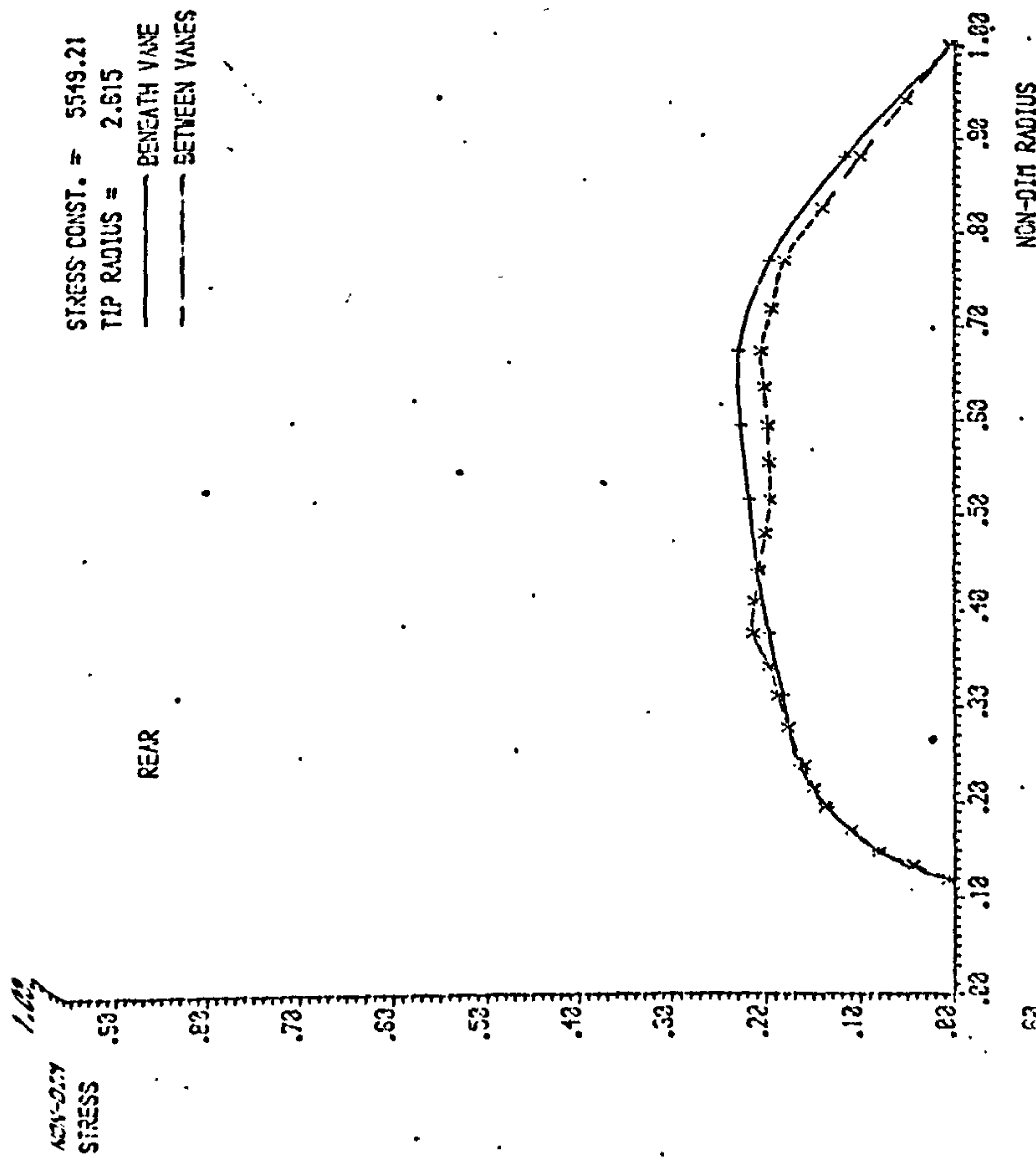
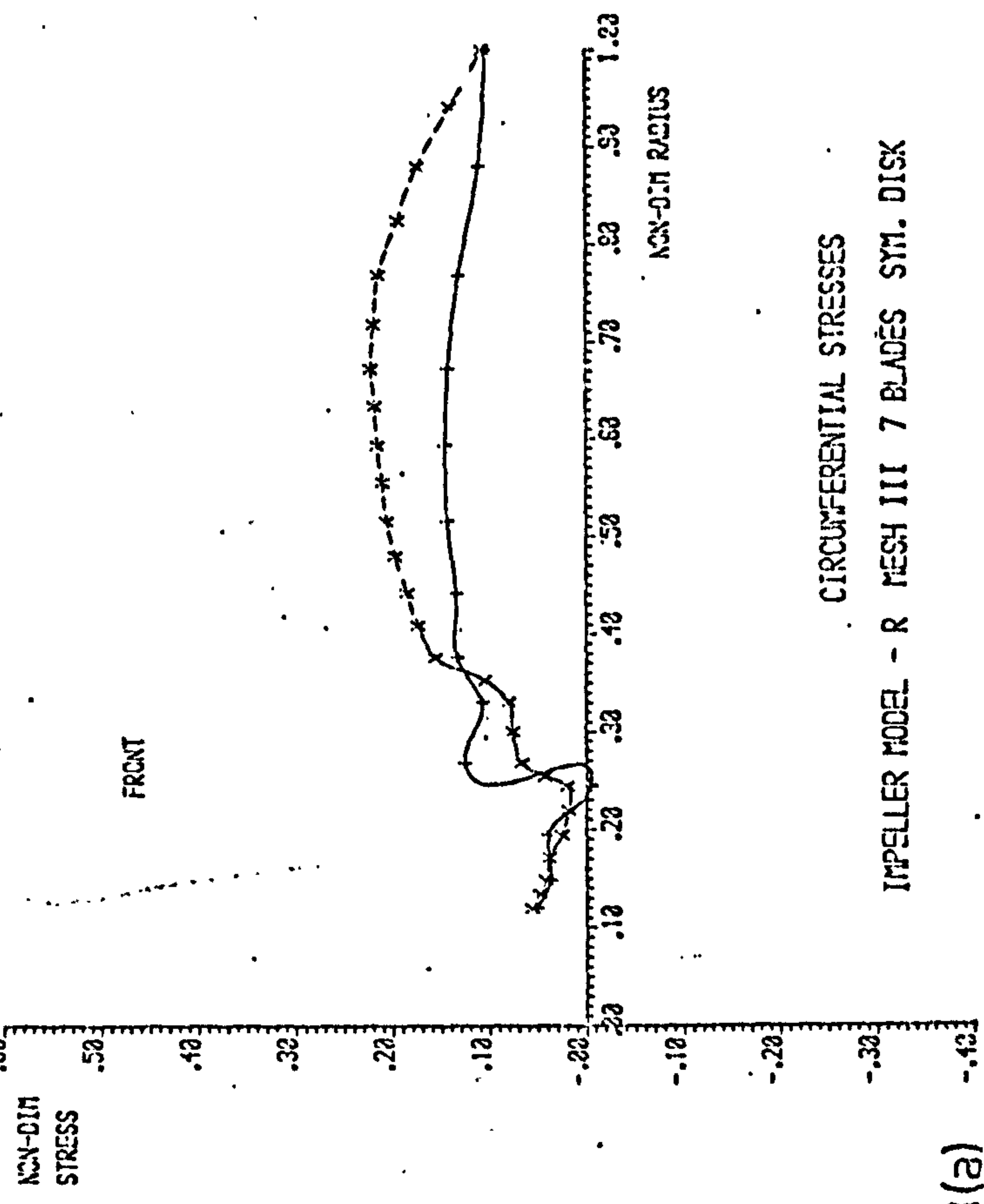
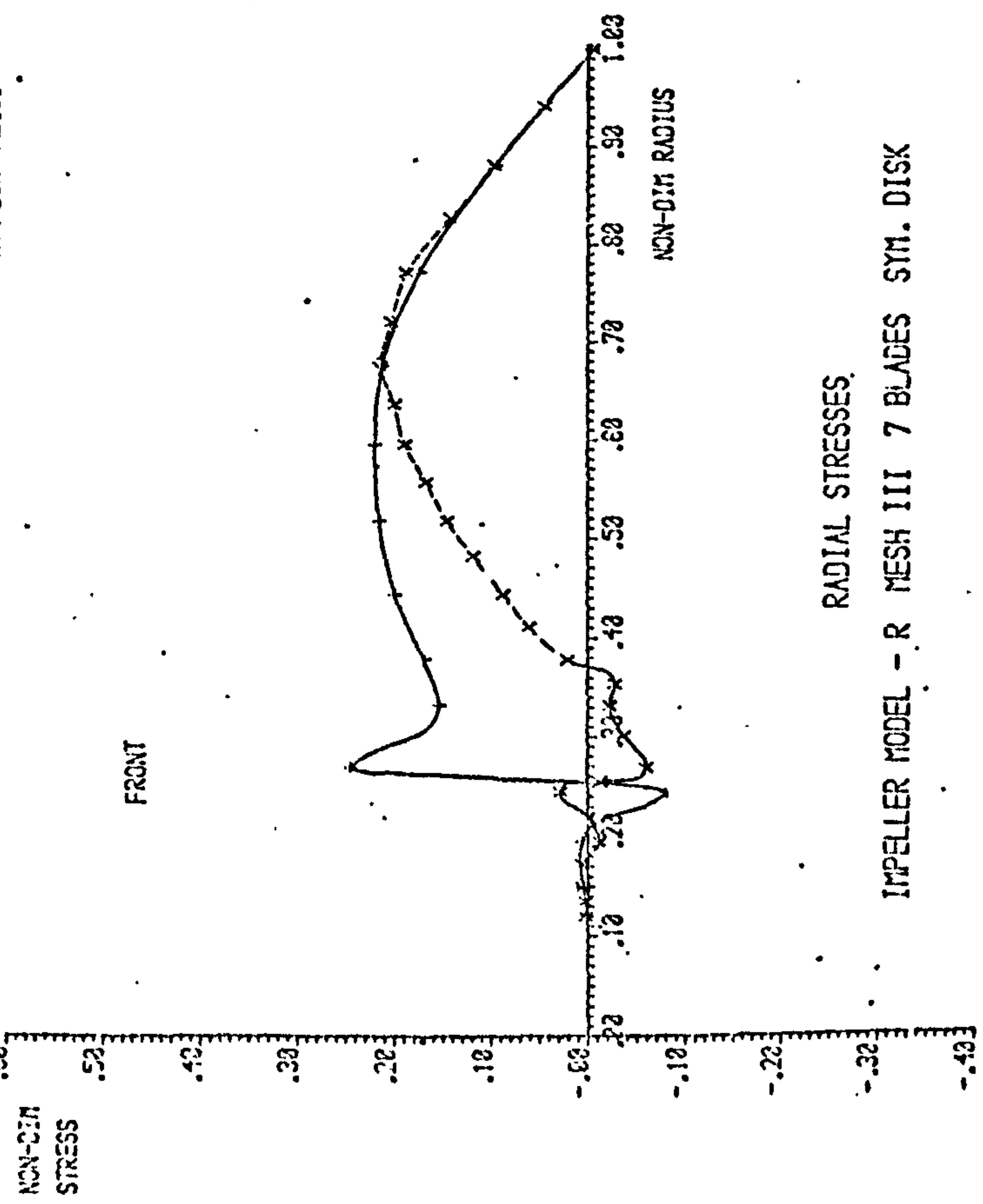
AXIAL DISPLACEMENTS
IMPELLER MODEL - P WITH COVER DISK MESH III

STRESS CONST. = 5549.21
TIP RADIUS = 2.615
TIP AXIS = 1.330



RADIAL STRESS ON BLADE LEADING EDGE
IMPELLER MODEL - P WITH COVER DISK MESH III

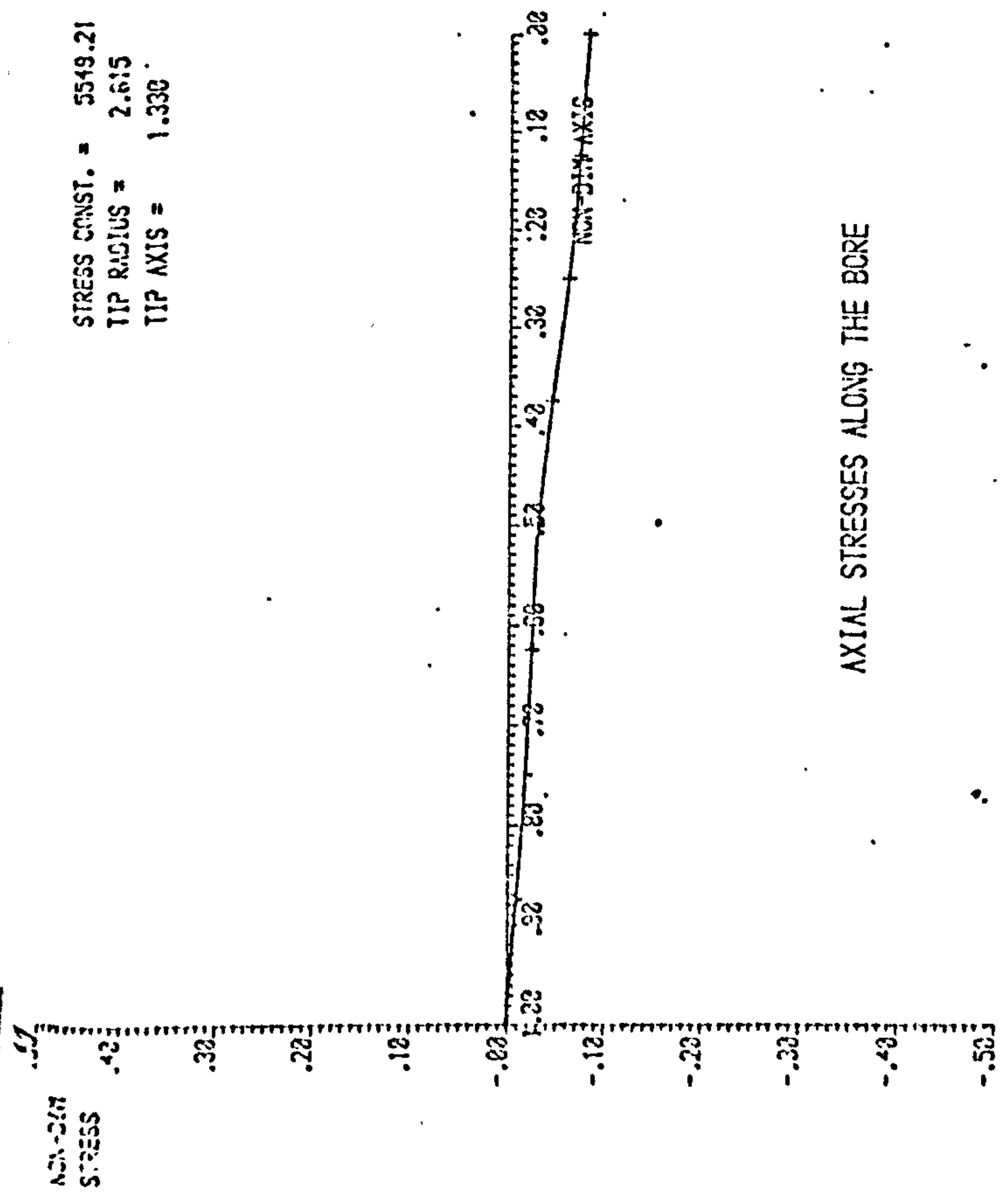
FIG 4-22(b)



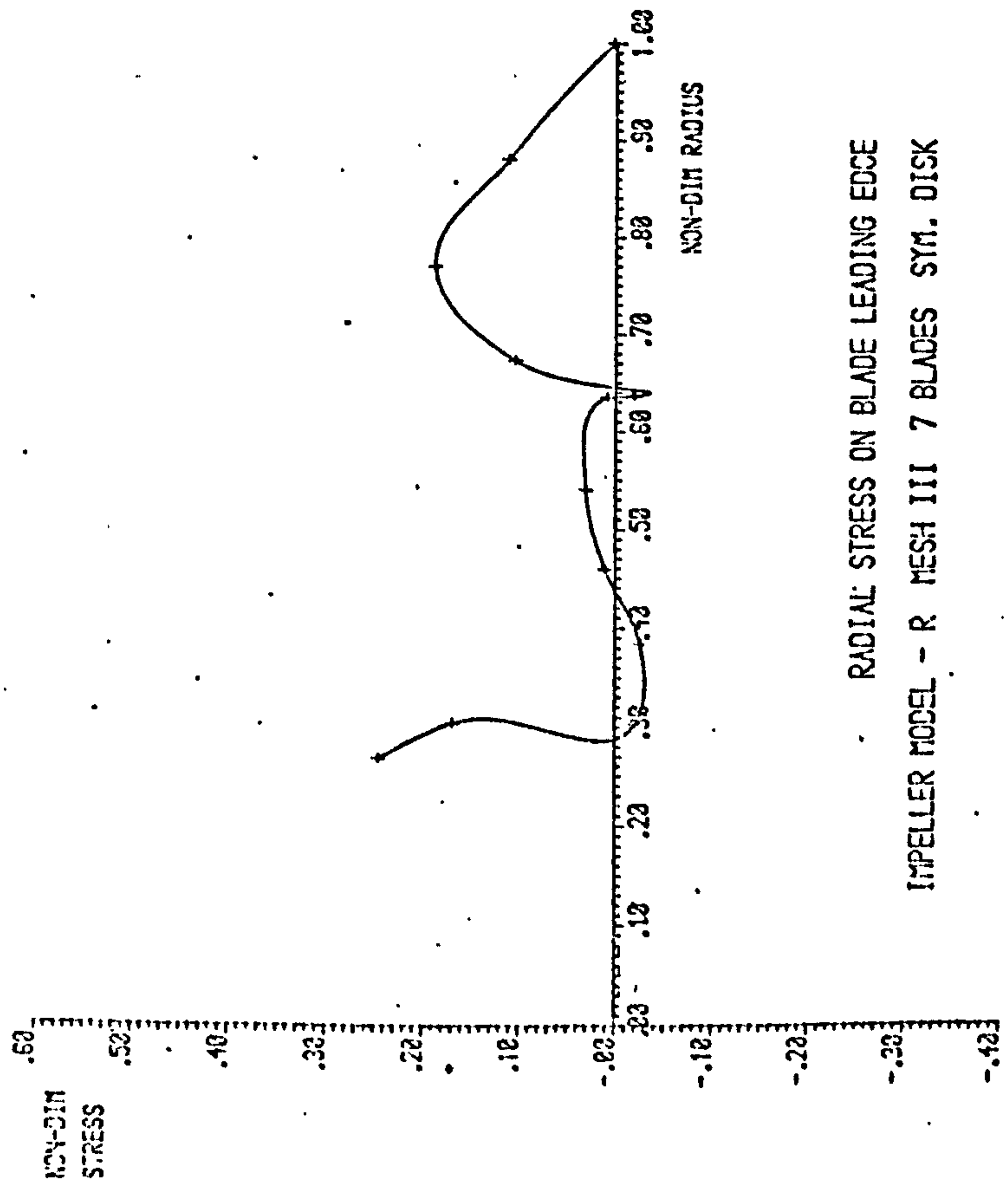
CIRCUMFERENTIAL STRESSES
IMPELLER MODEL - R MESH III 7 BLADES SYM. DISK

RADIAL STRESSES
IMPELLER MODEL - R MESH III 7 BLADES SYM. DISK

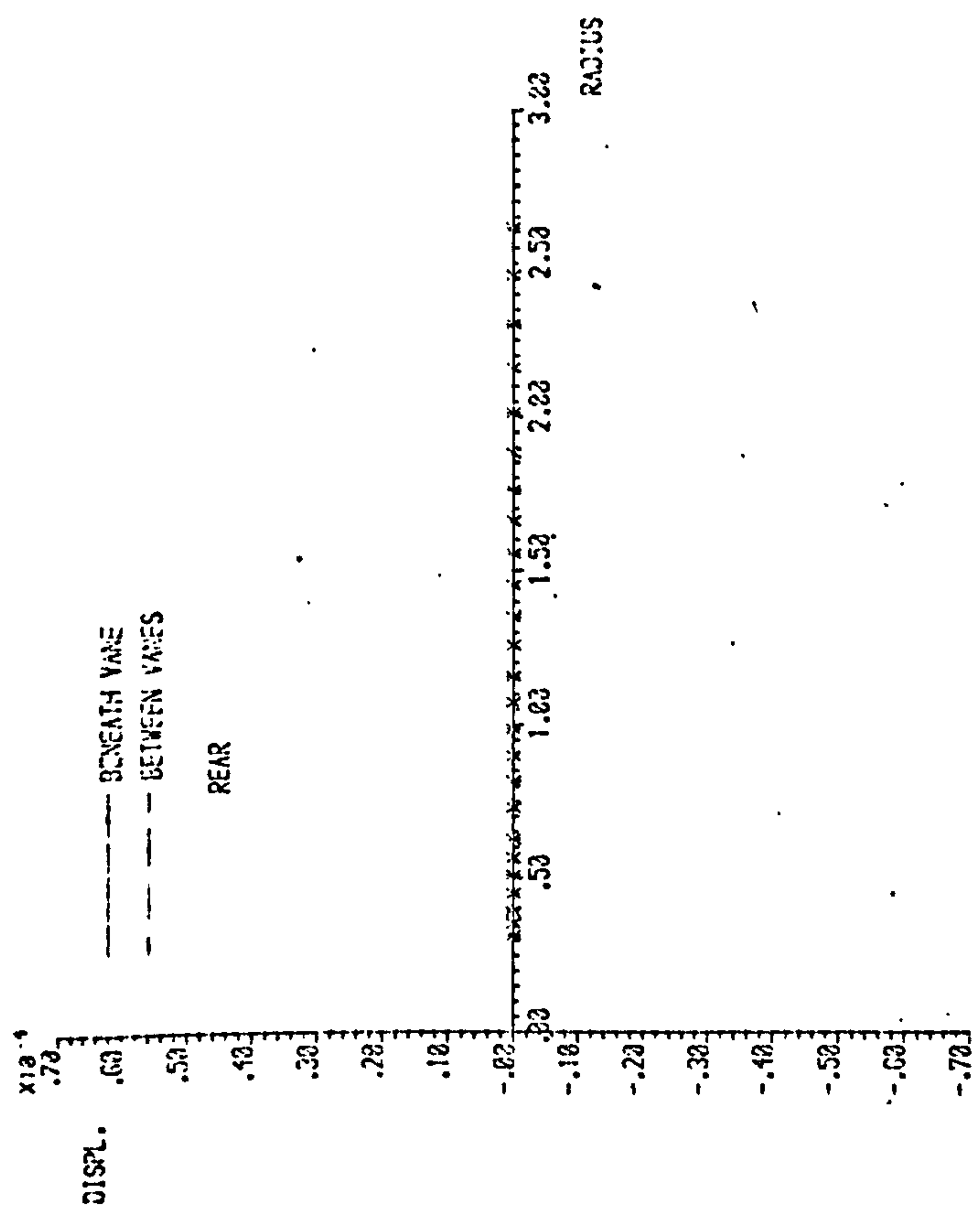
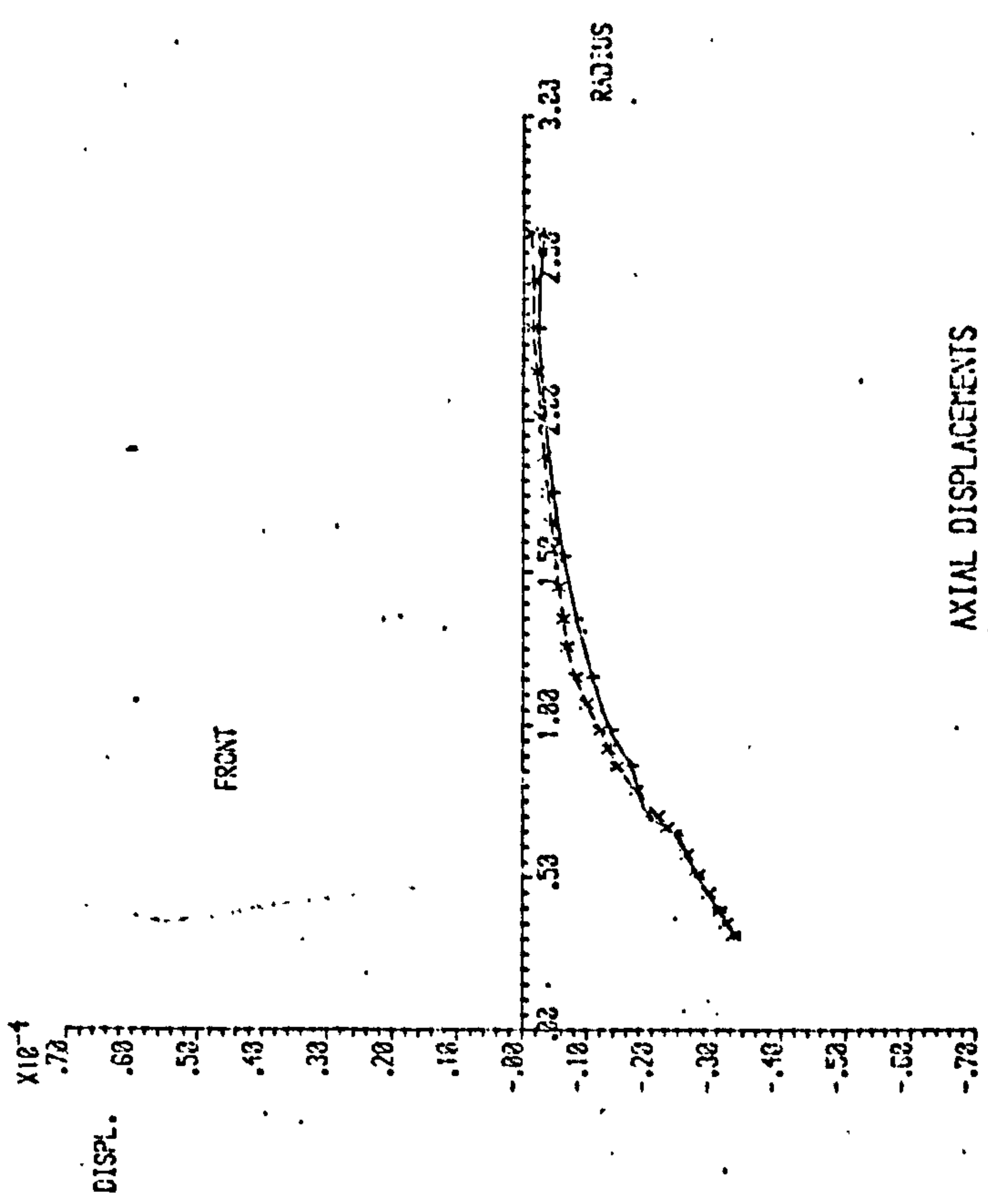
FIG 4-23(a)



AXIAL STRESSES ALONG THE BORE



RADIAL STRESS ON BLADE LEADING EDGE
IMPELLER MODEL - R MESH III 7 BLADES SYM. DISK



AXIAL DISPLACEMENTS
IMPELLER MODEL - R MESH III 7 BLADES SYM. DISK

FIG 4-23(b)

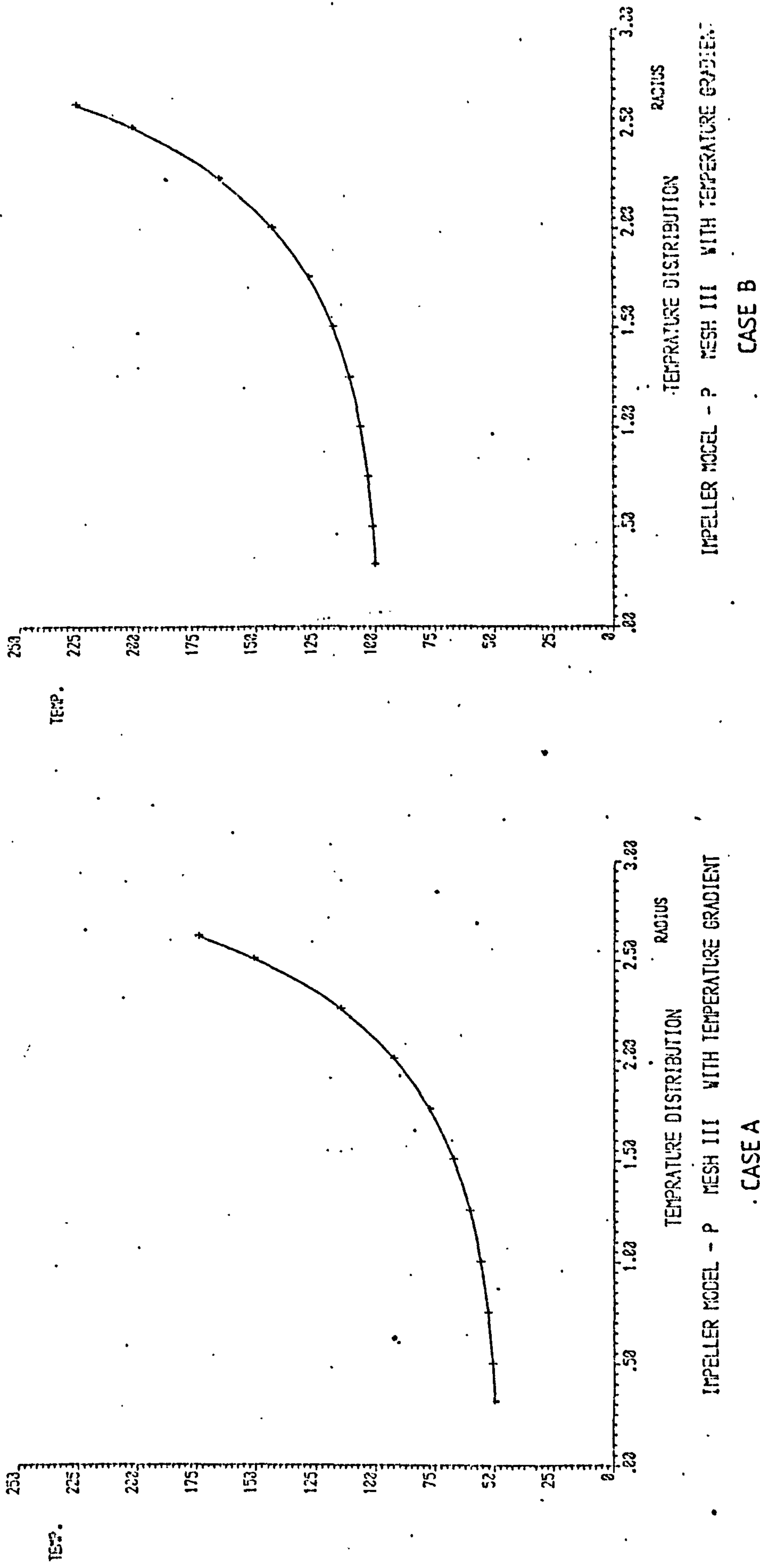
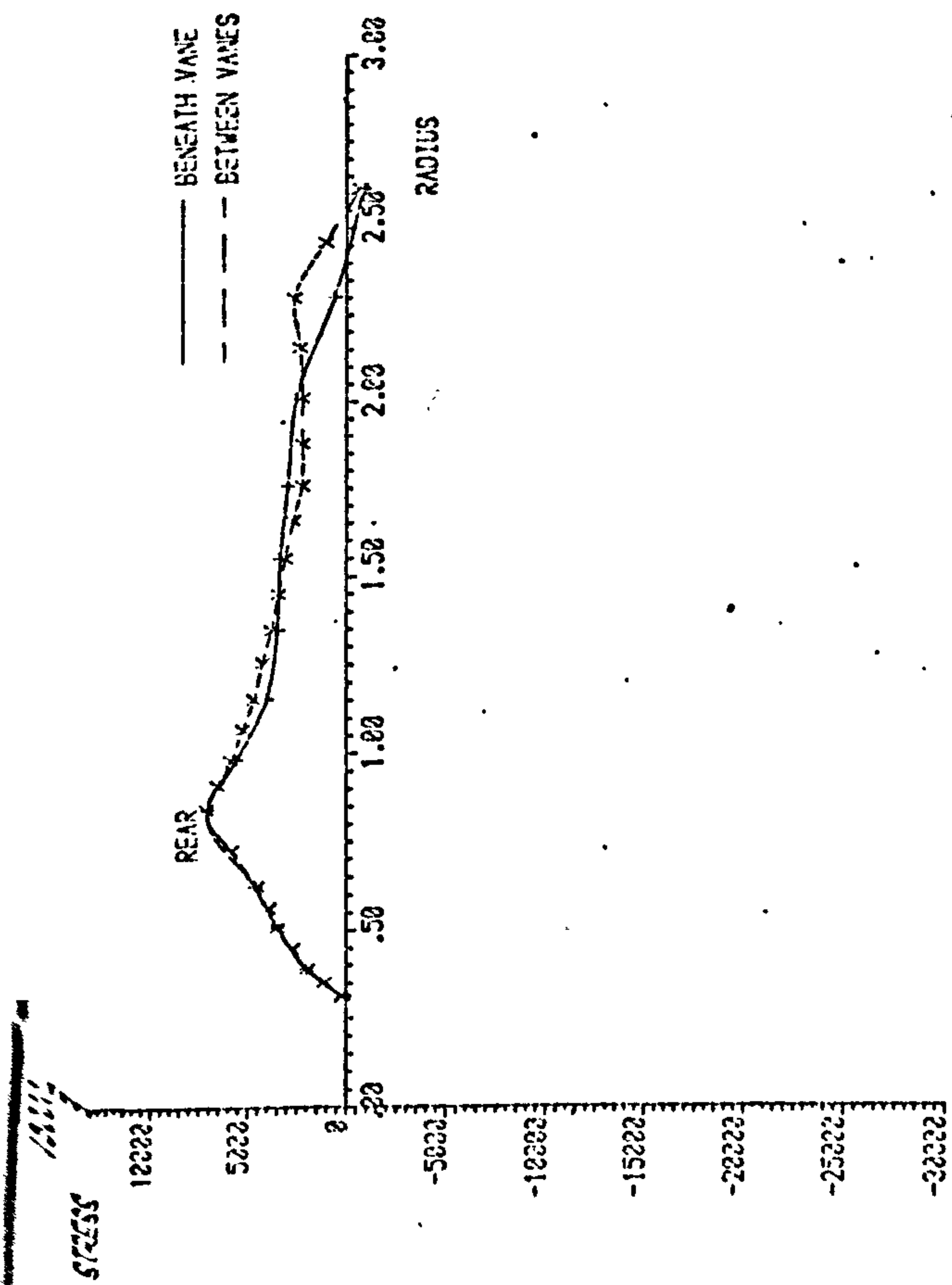
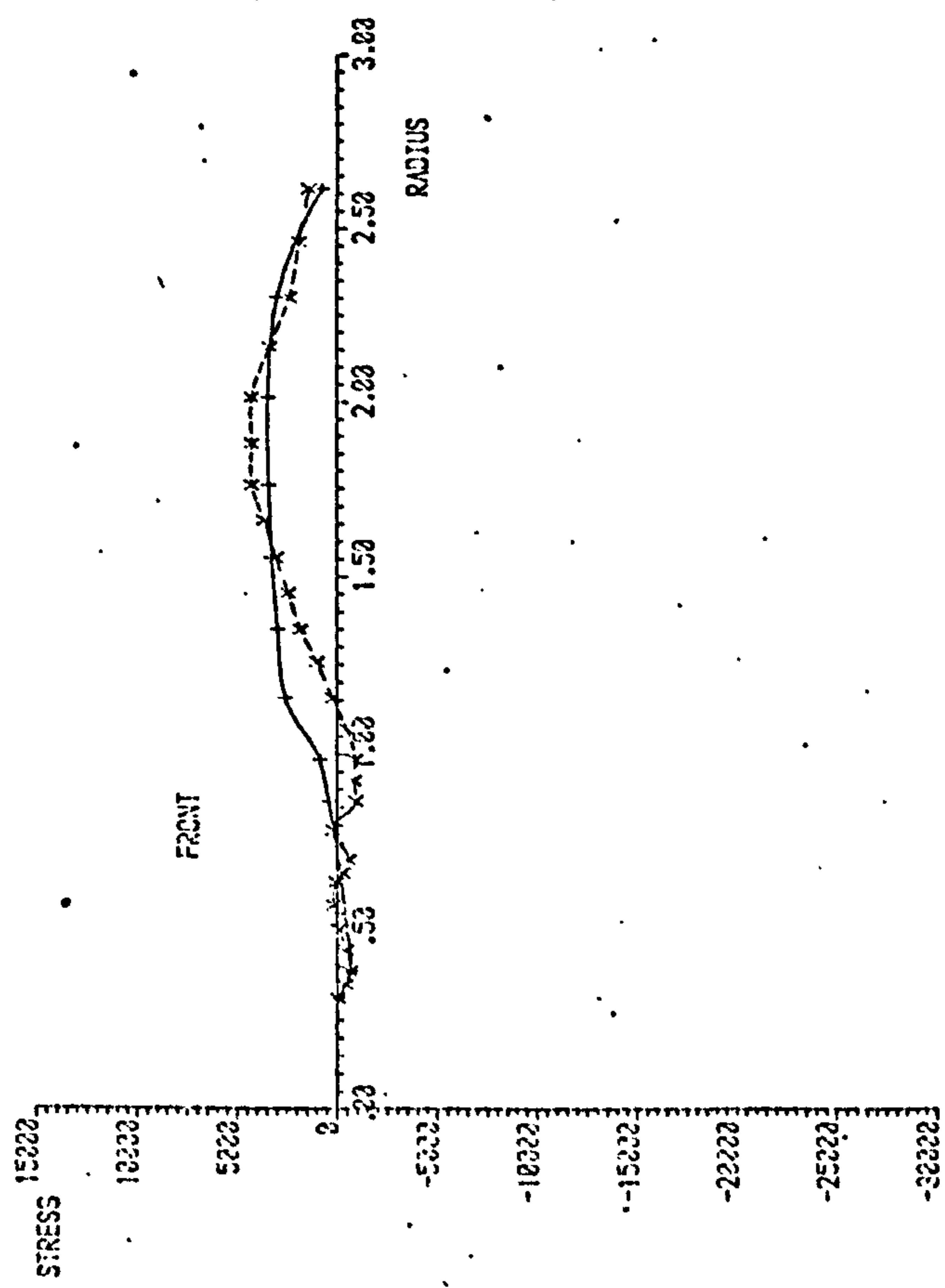
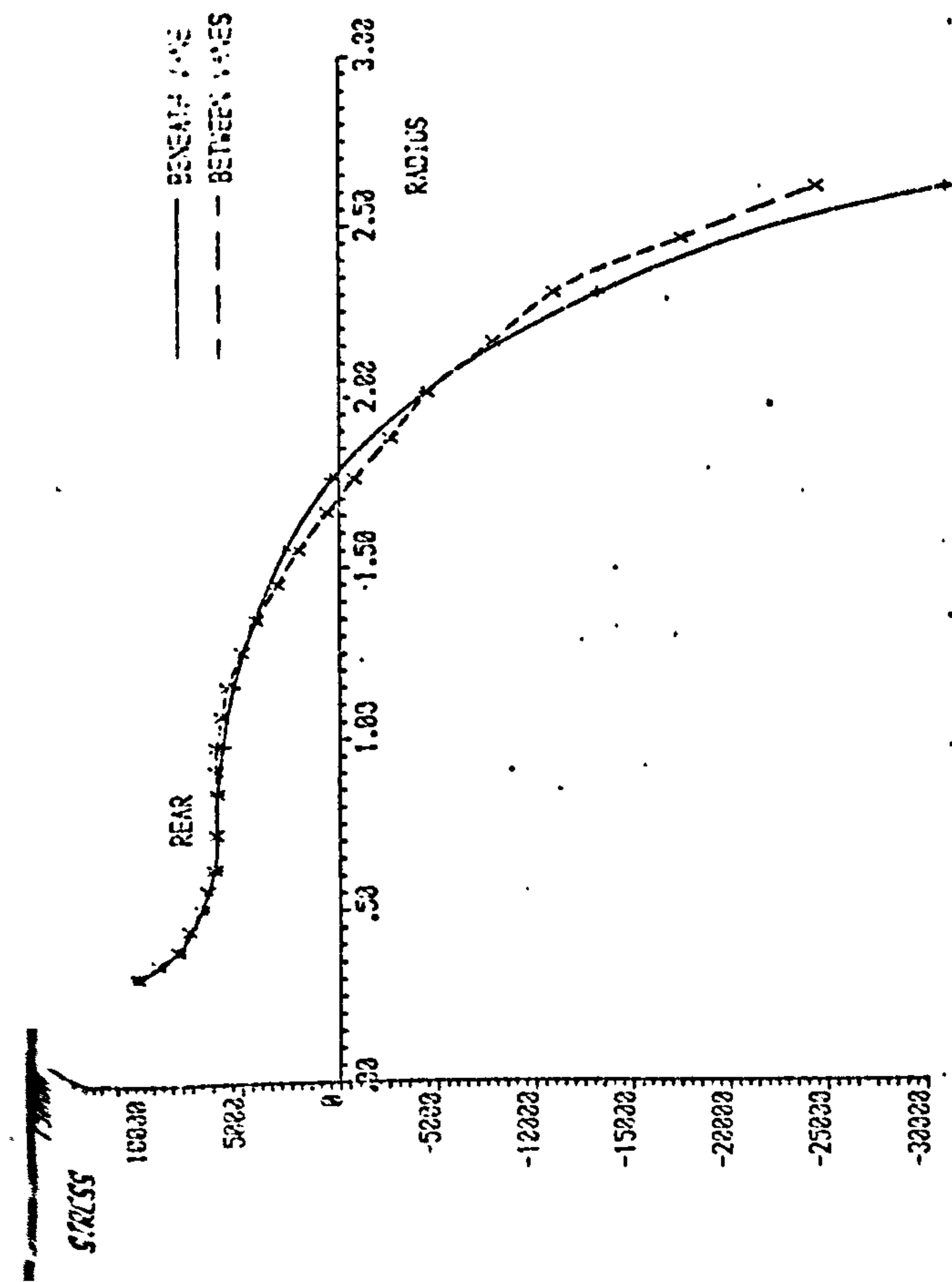
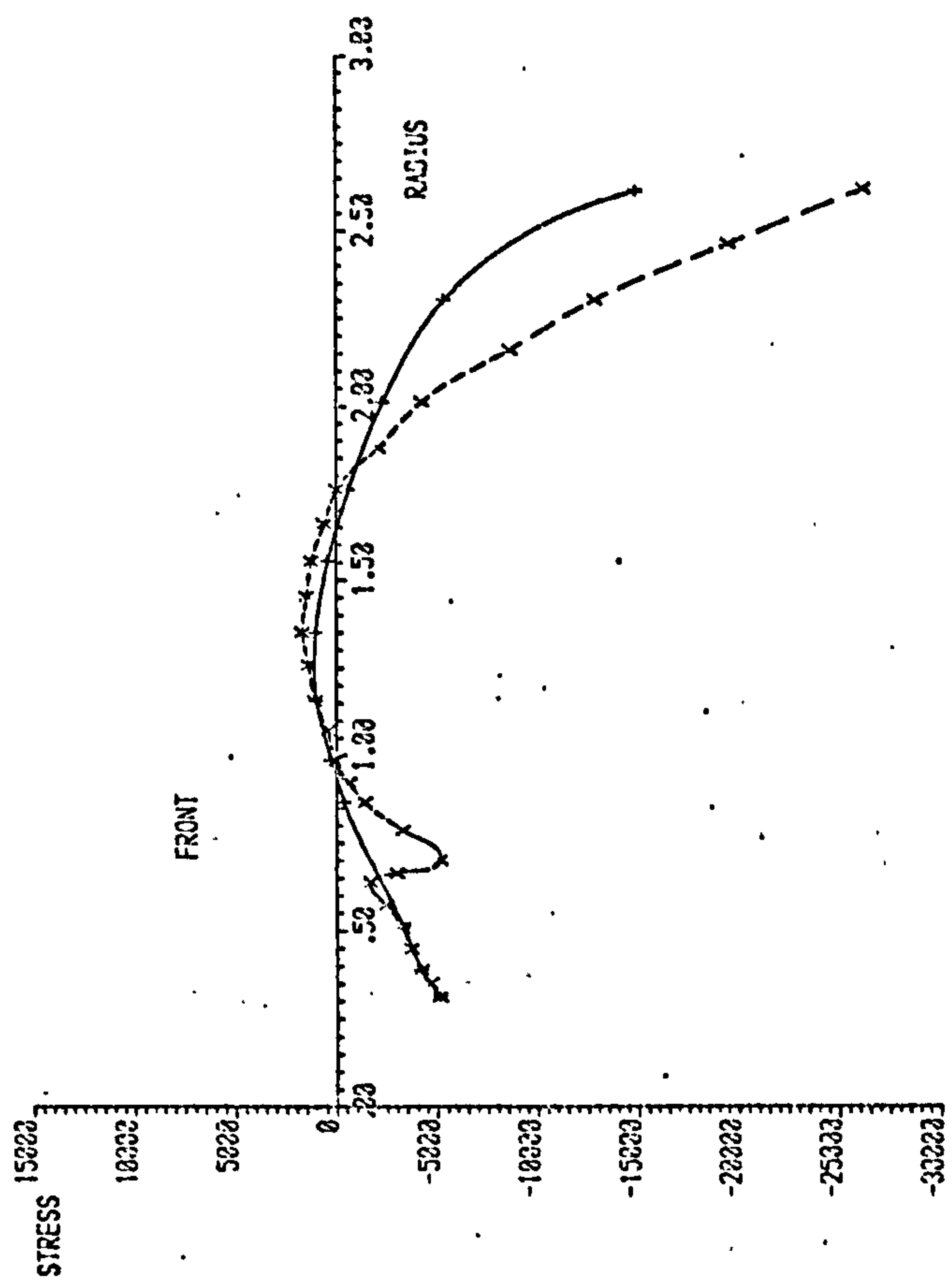


FIG 4-24(a)



RADIAL STRESSES

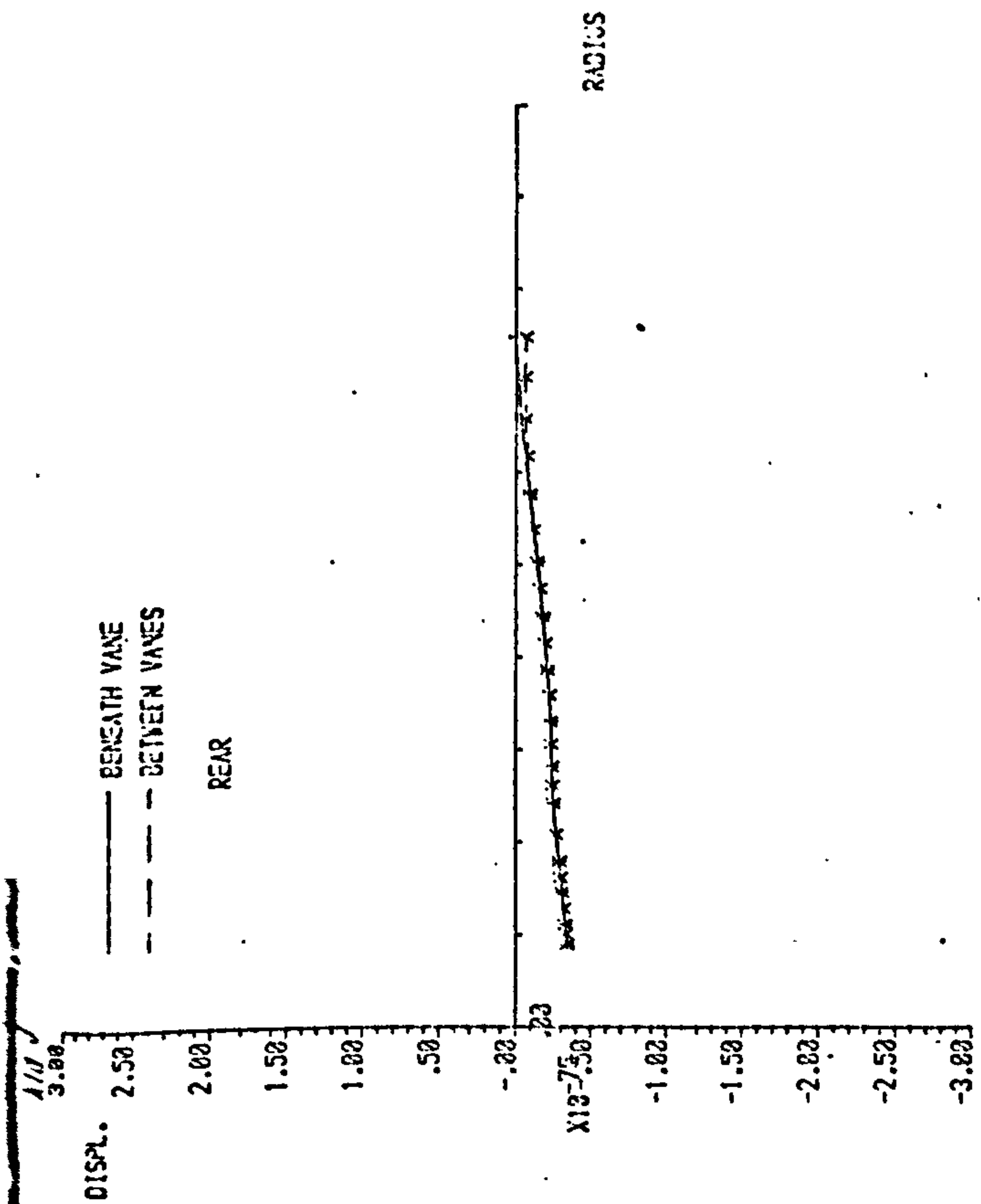
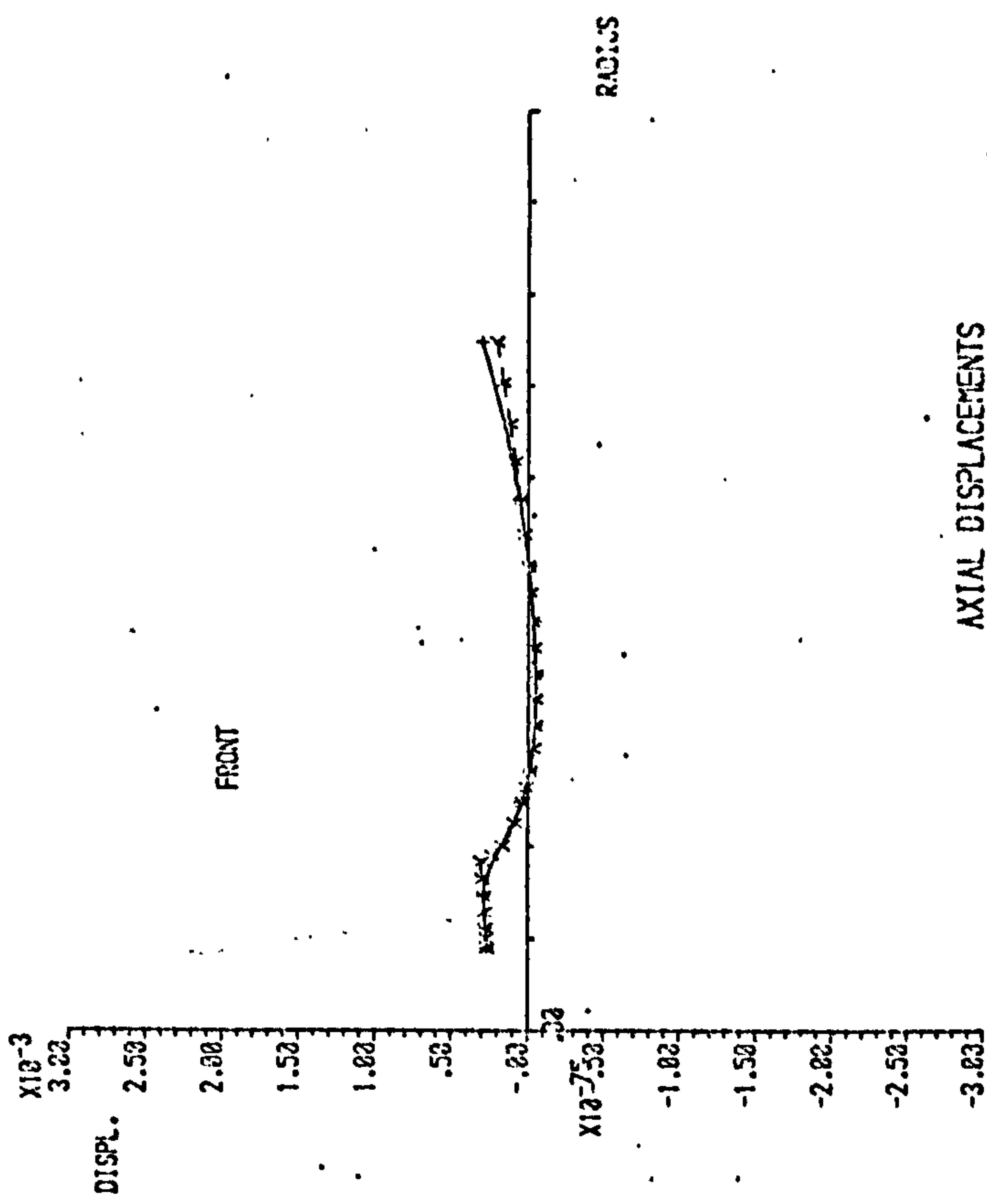
IMPELLER MODEL - P MESH III WITH TEMPERATURE GRADIENT



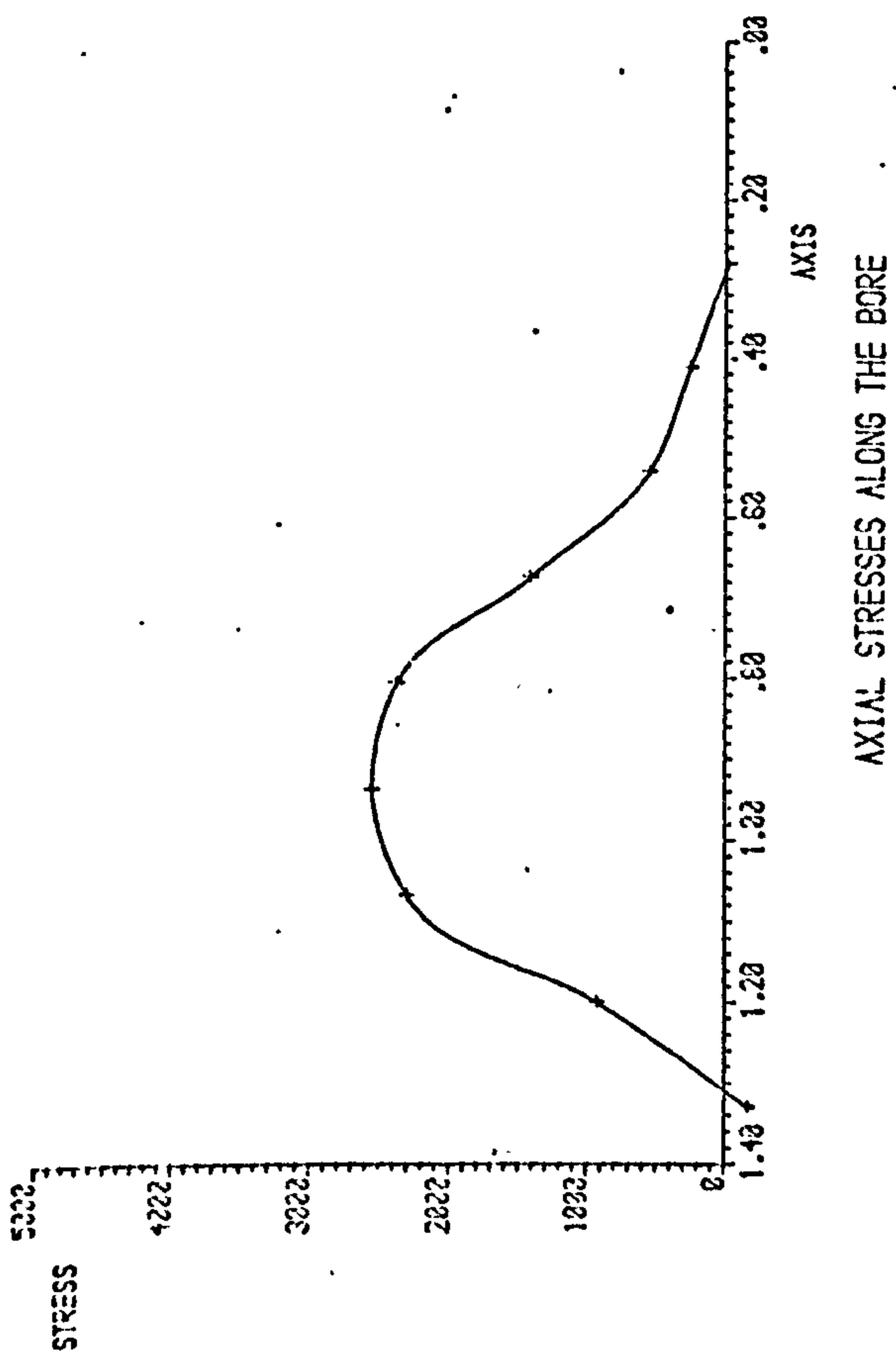
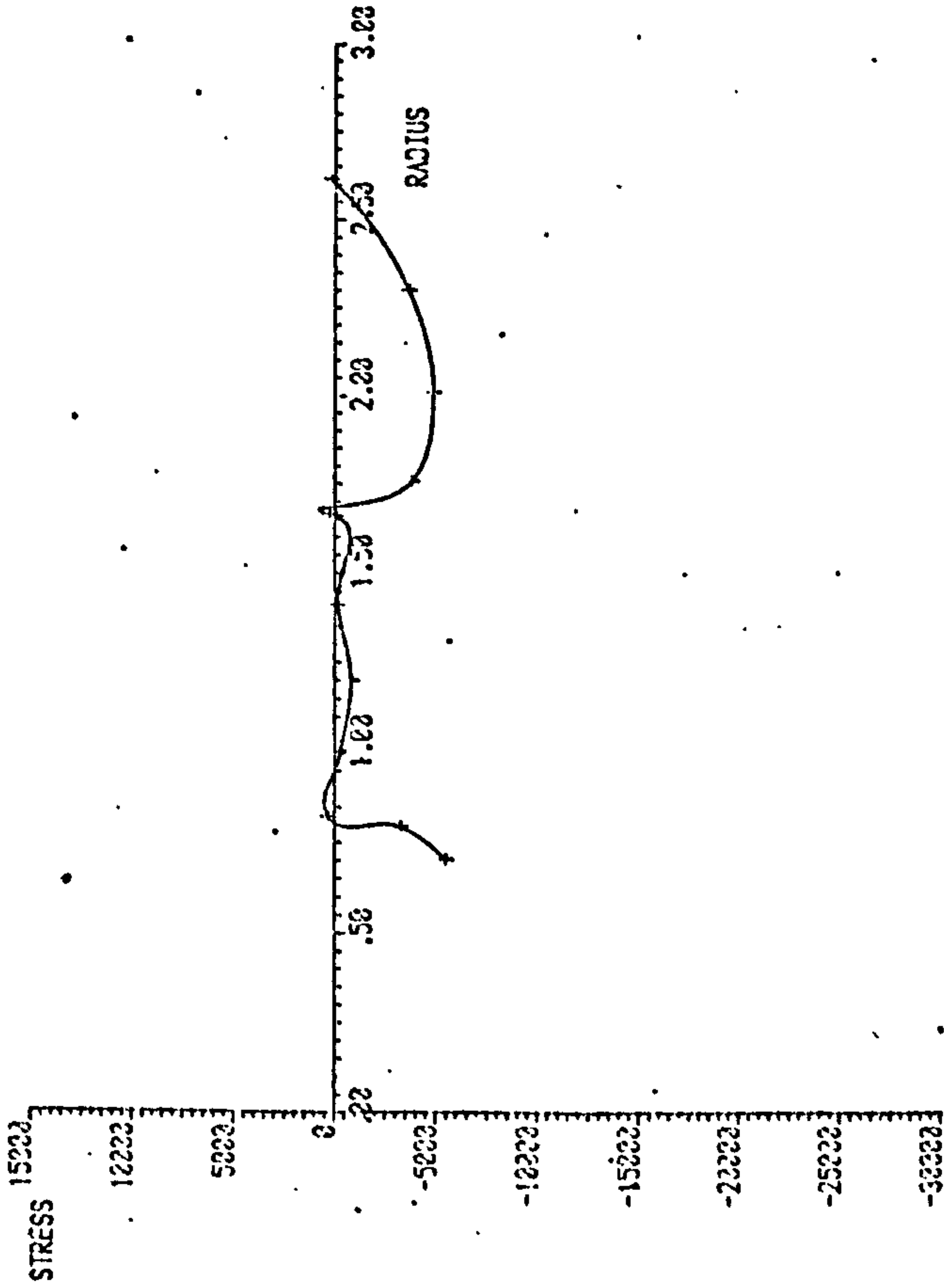
CIRCUMFERENTIAL STRESSES

IMPELLER MODEL - P MESH III WITH TEMPERATURE GRADIENT

FIG 4-24(b)

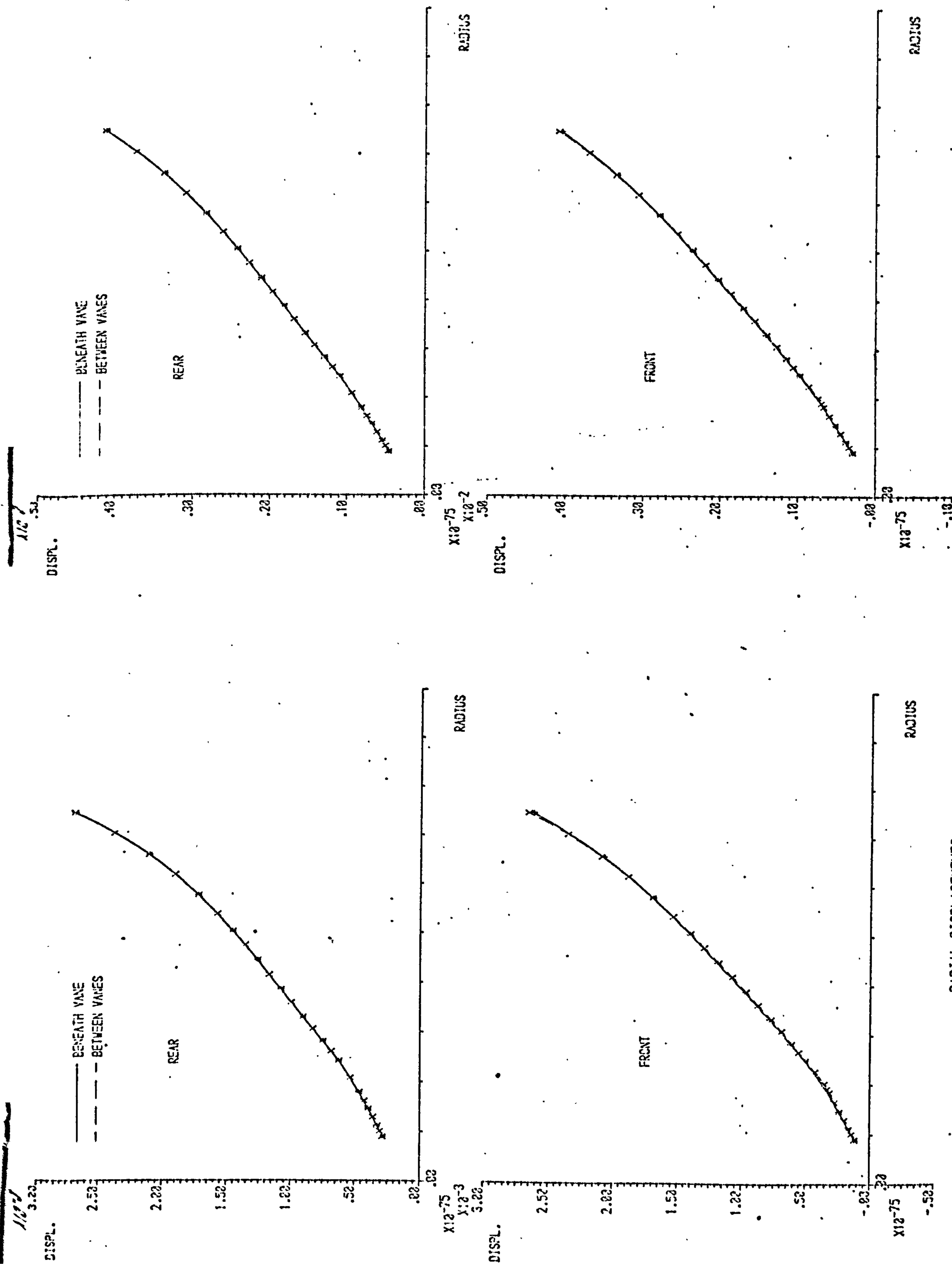


AXIAL DISPLACEMENTS
IMPELLER MODEL - P MESH III WITH TEMPERATURE GRADIENT



RADIAL STRESS ON BLADE LEADING EDGE
IMPELLER MODEL - P MESH III WITH TEMPERATURE GRADIENT

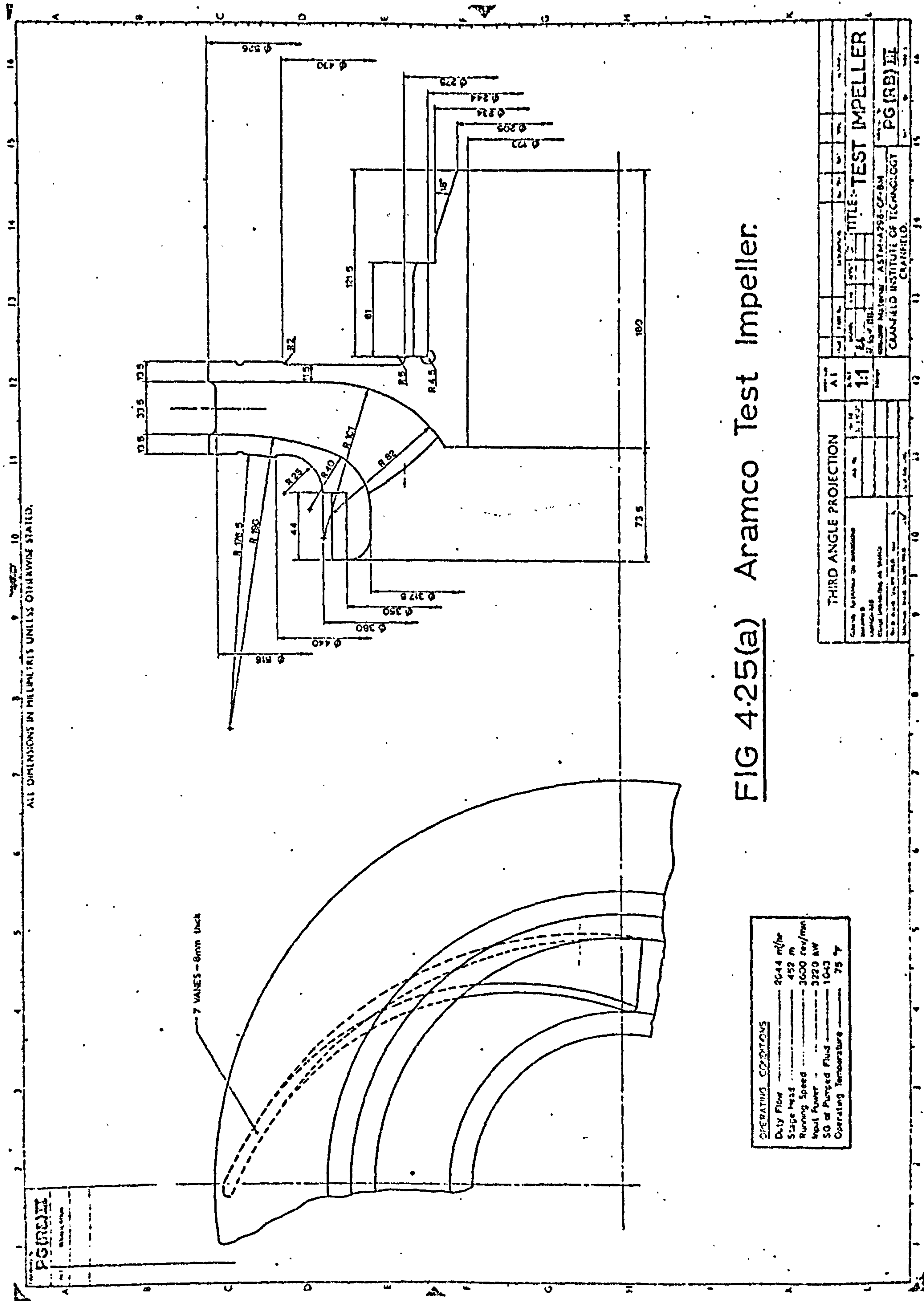
FIG 4-24(c)



RADIAL DISPLACEMENTS
 IMPELLER MODEL - P MESH III WITH TEMPERATURE GRADIENT
 CASE B

FIG 4-24(d)

RADIAL DISPLACEMENTS
 IMPELLER MODEL - P MESH III WITH TEMPERATURE GRADIENT
 CASE A



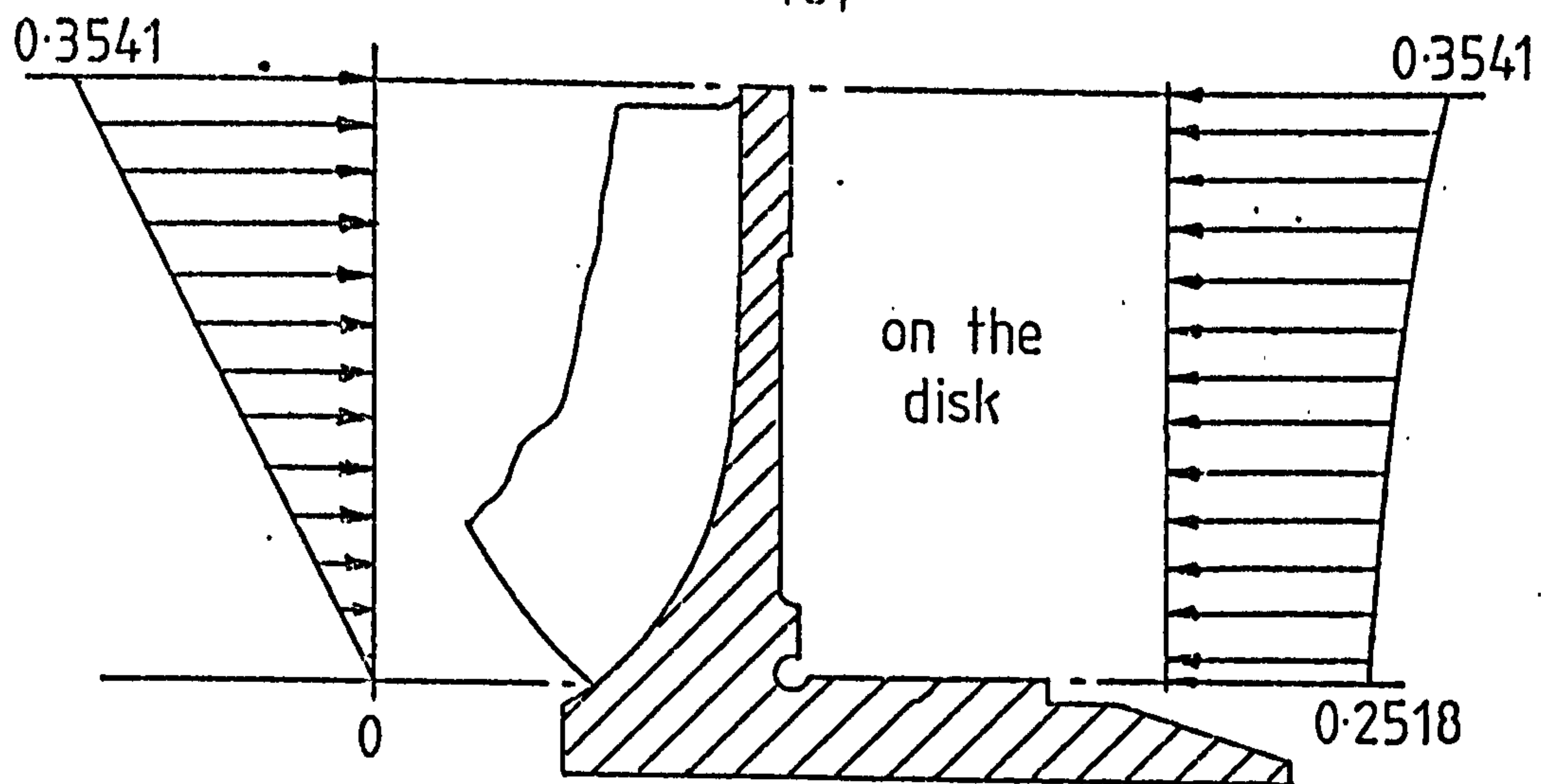
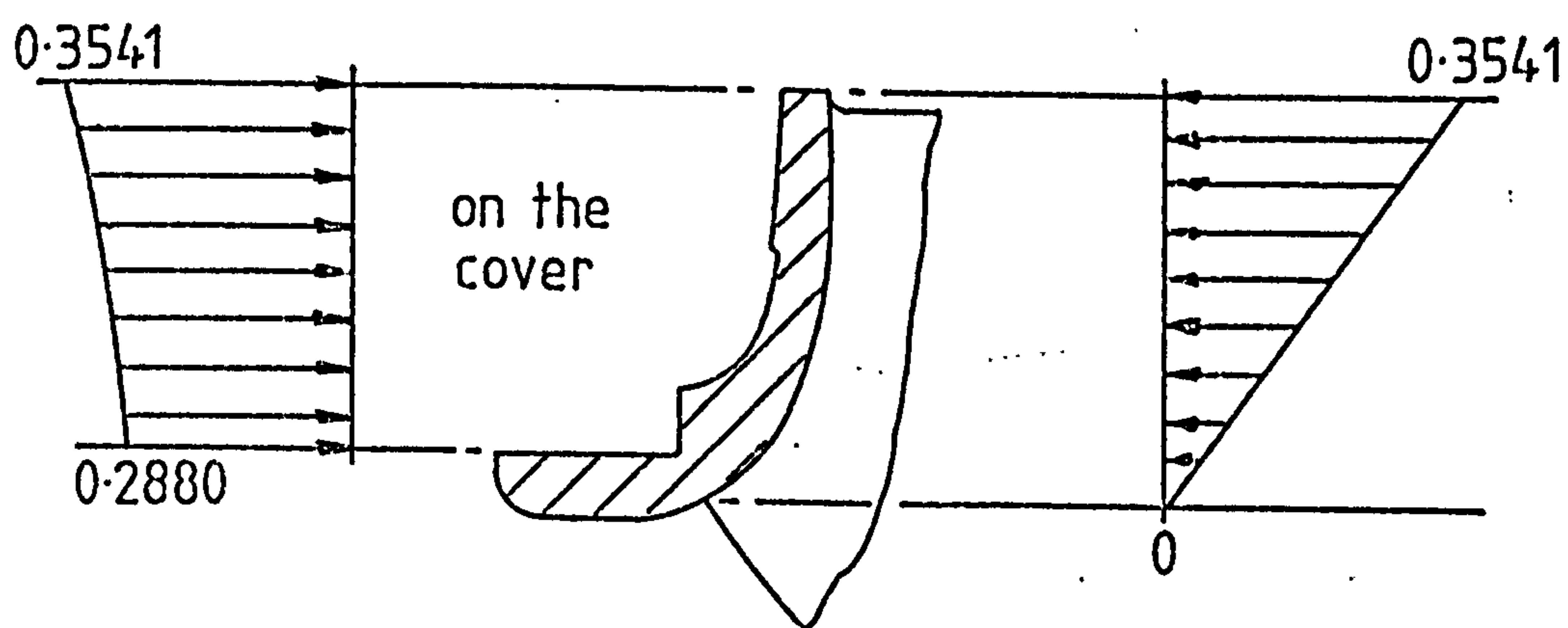
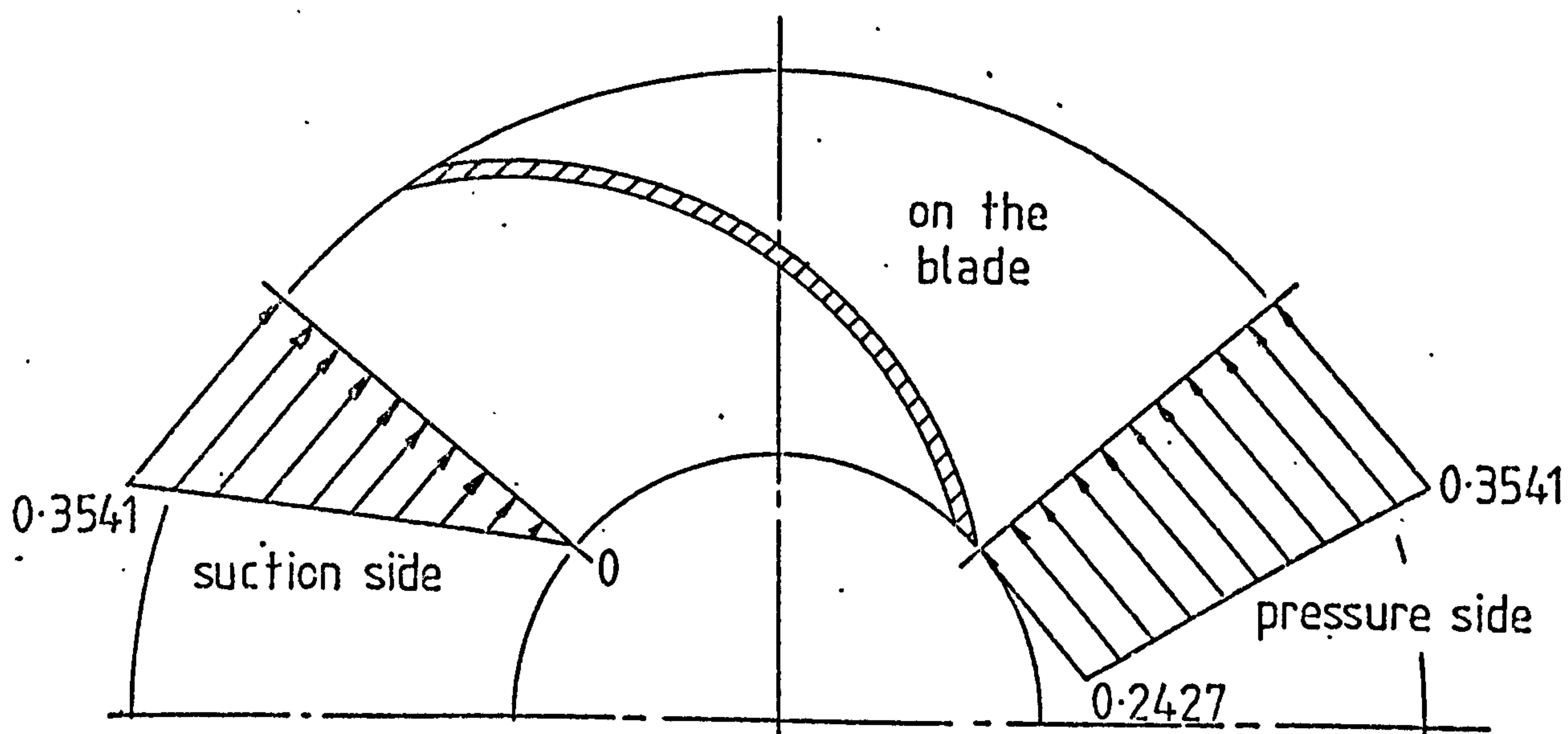
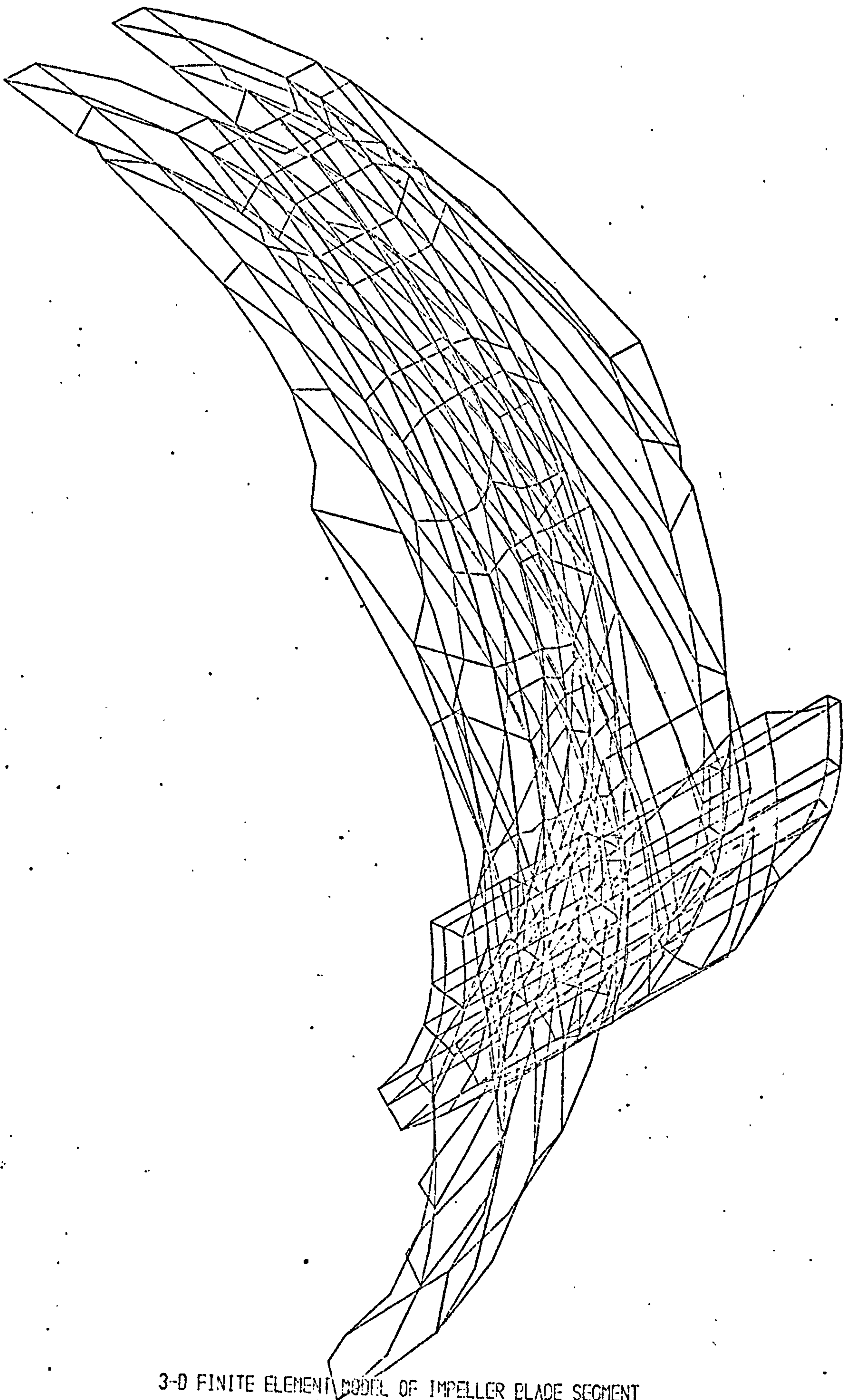


FIG 4.25(b) Fluid Pressure Distribution.



3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN MM SCALE: .50 TIMES THE SUPPLIED UNITS

FIG 4-26(a) ARAMCO TEST IMPELLER

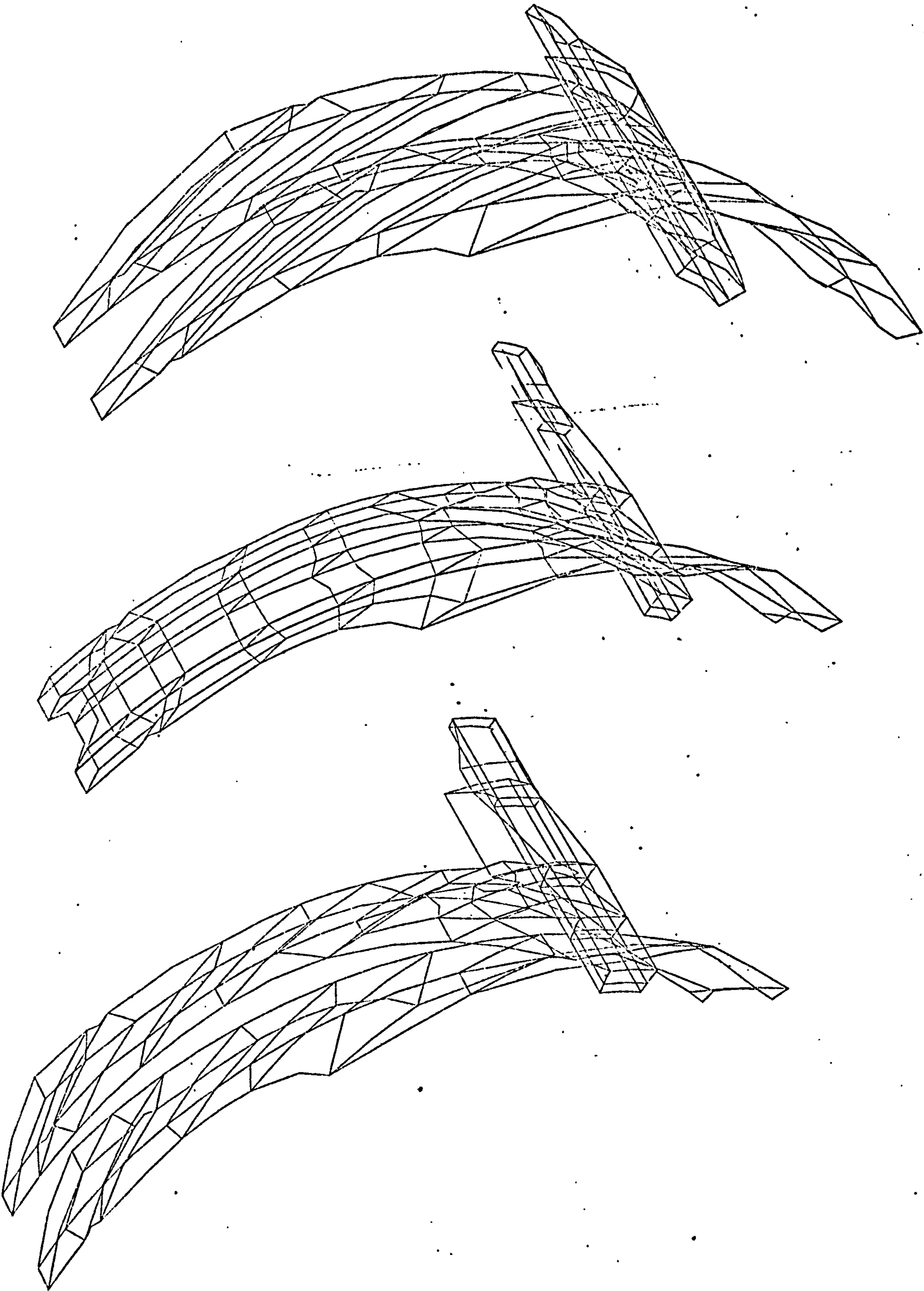
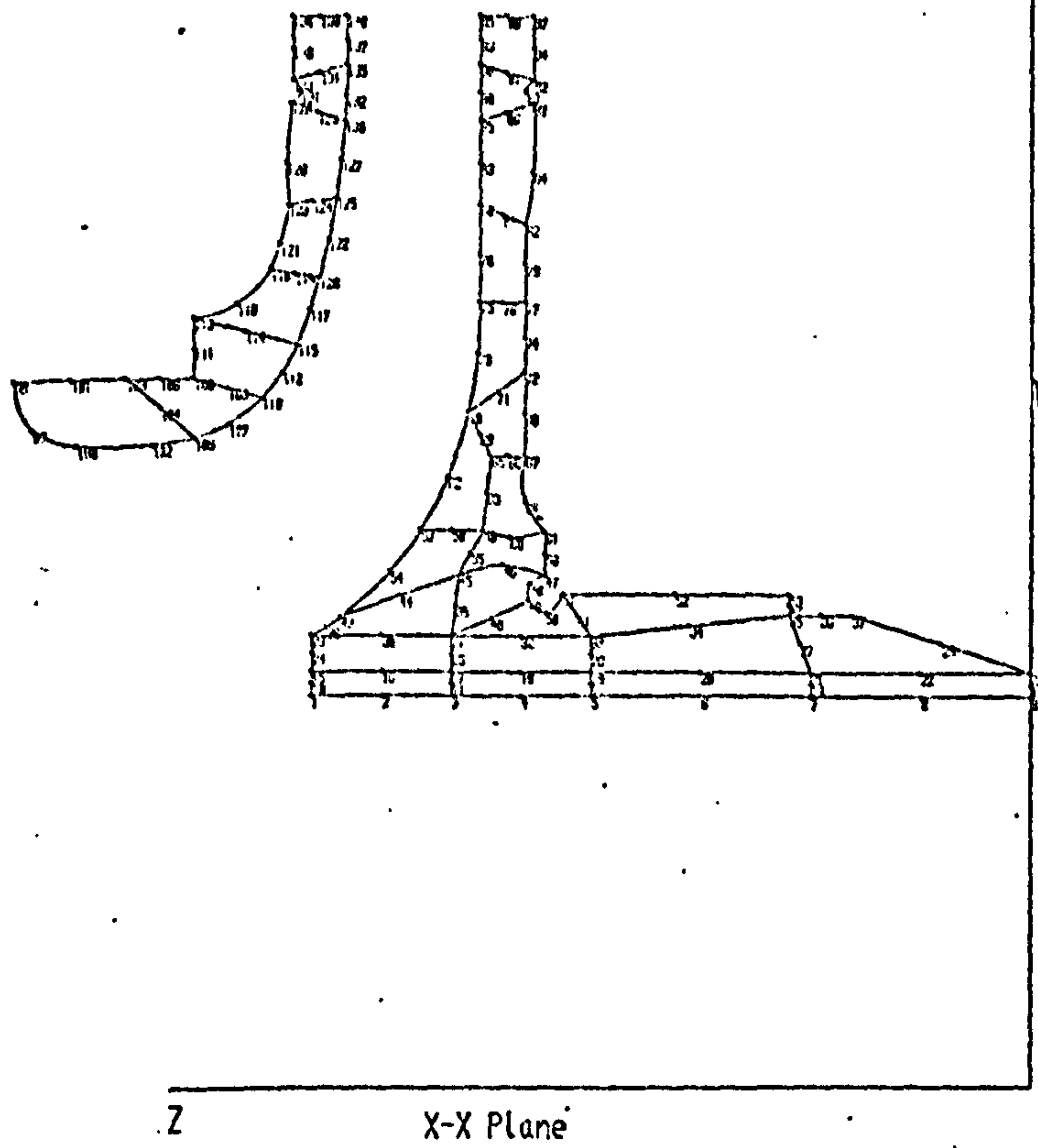
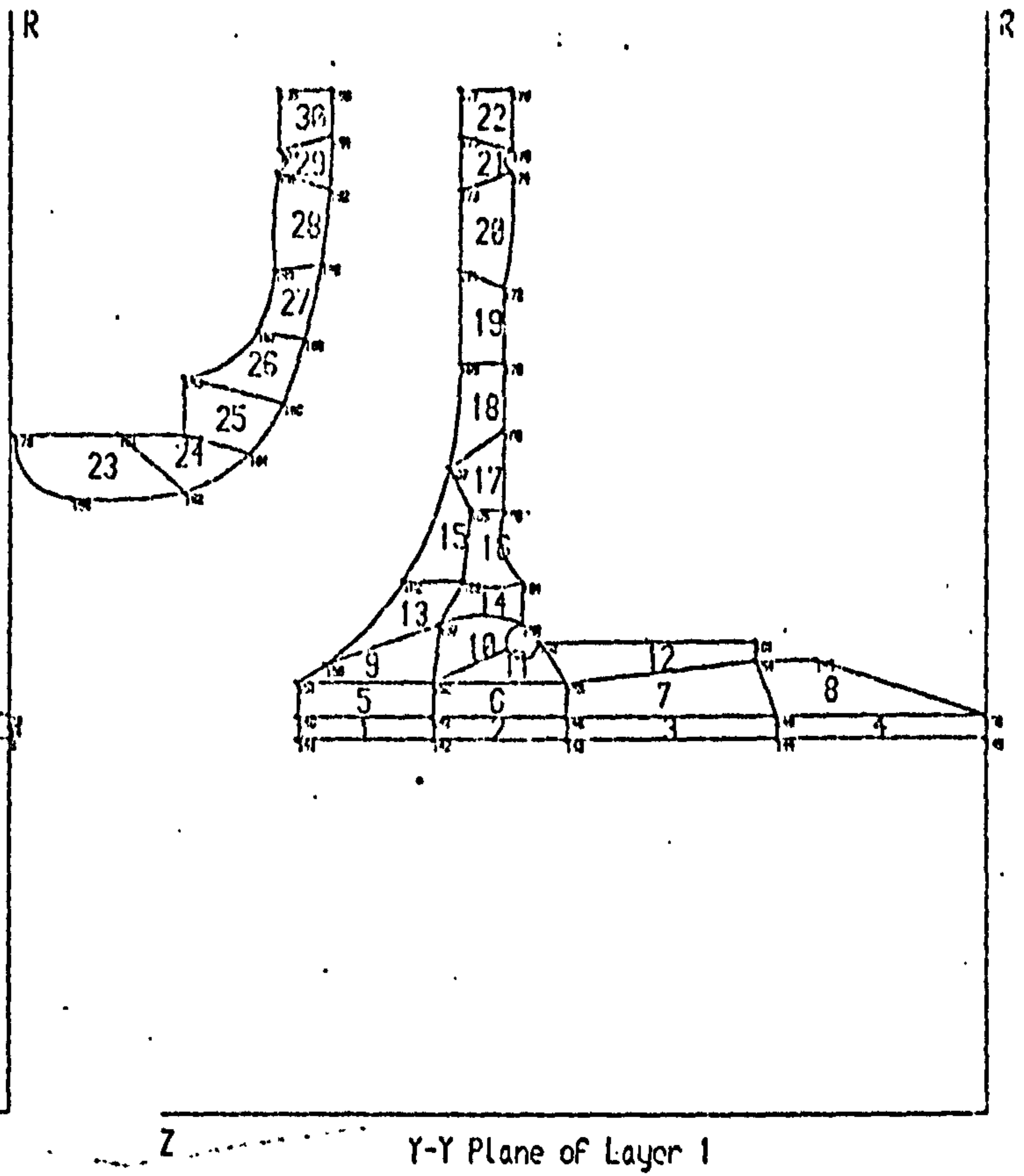


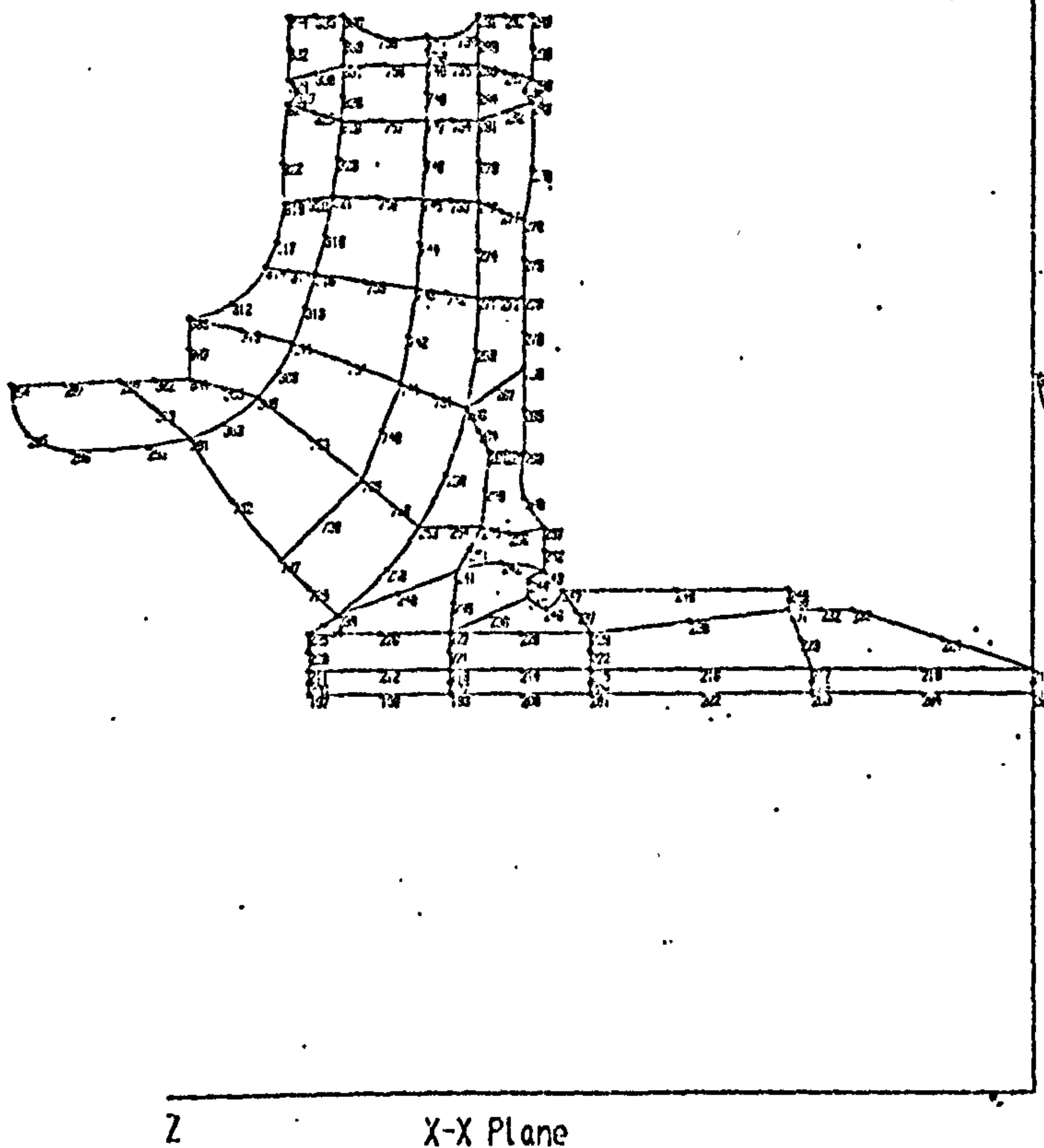
FIG 4-26 (b) An Exploded View.



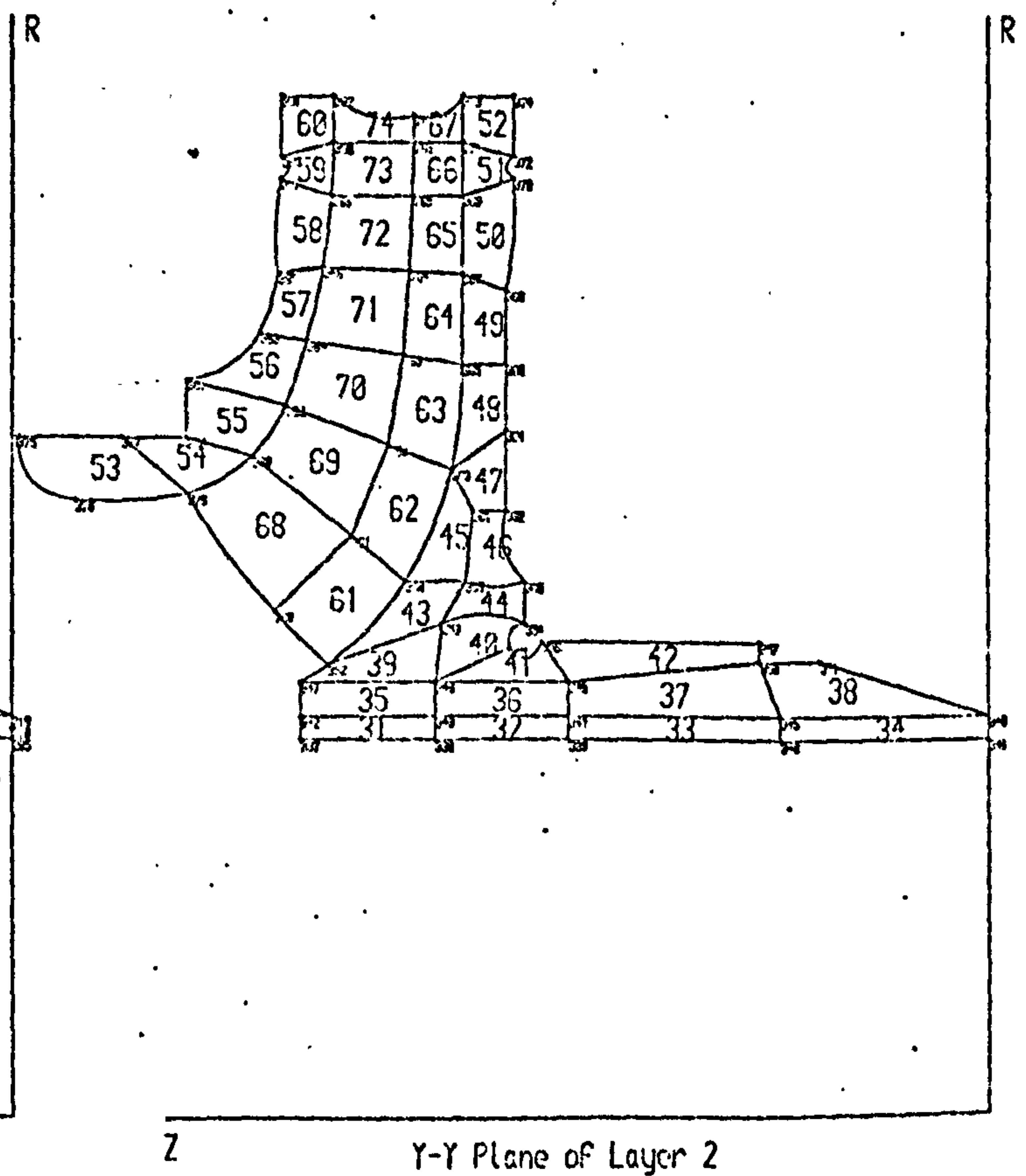
ARAMCO TEST IMPELLER



DRAWN IN MM SCALE: .50 TIMES THE SUPPLIED UNITS
Sheet 1

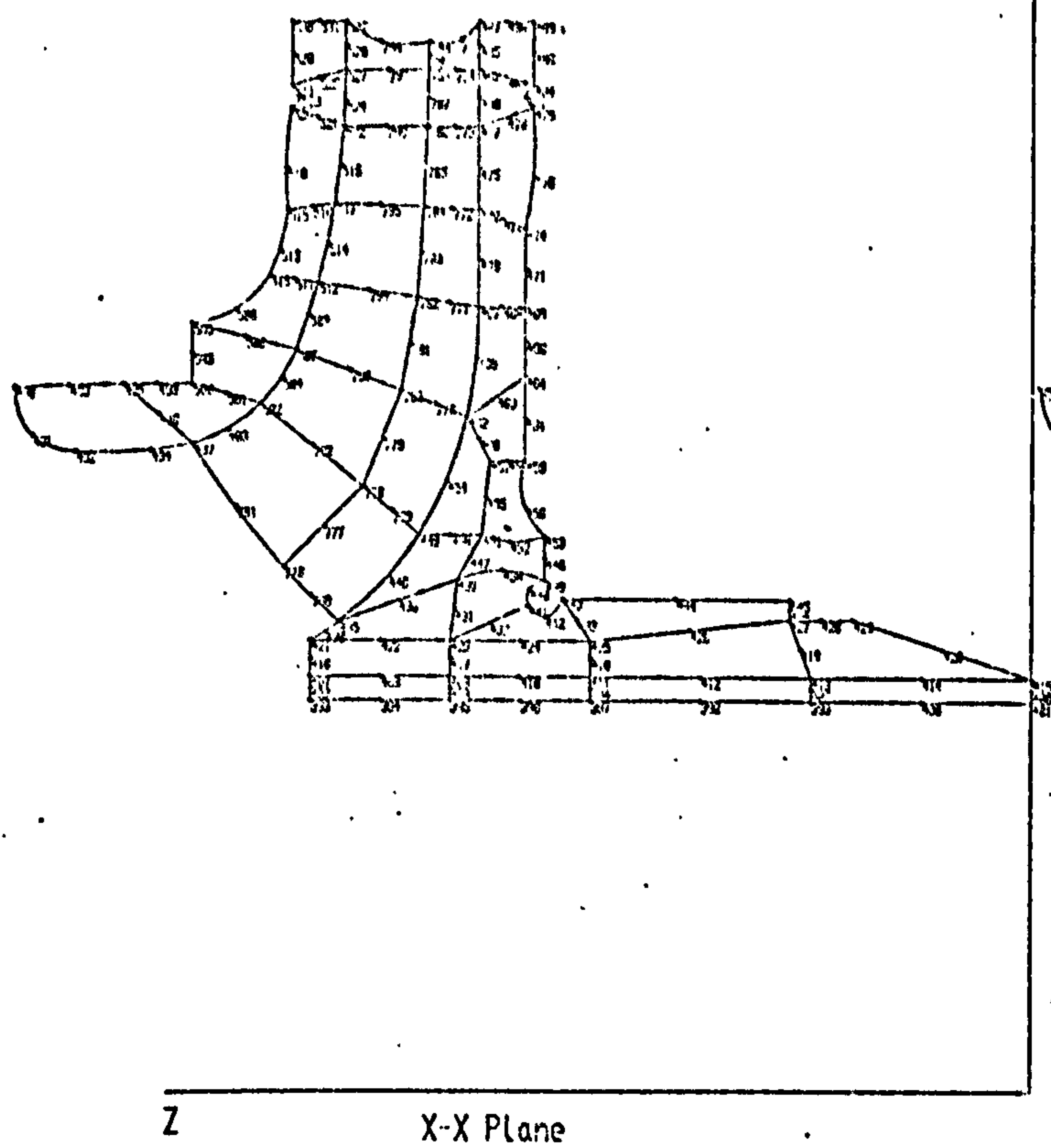


ARAMCO TEST IMPELLER

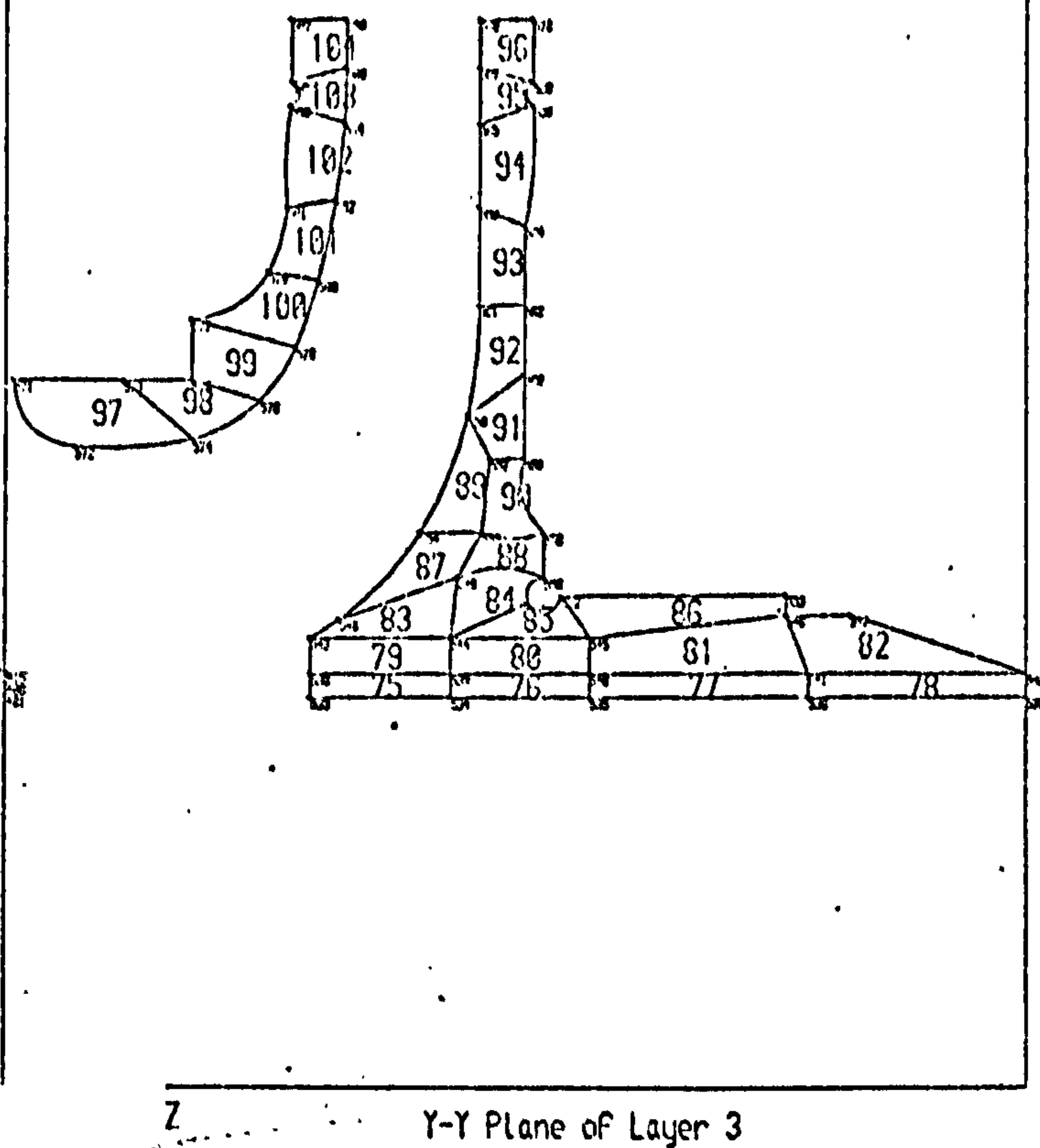


DRAWN IN MM SCALE: .50 TIMES THE SUPPLIED UNITS
Sheet 2

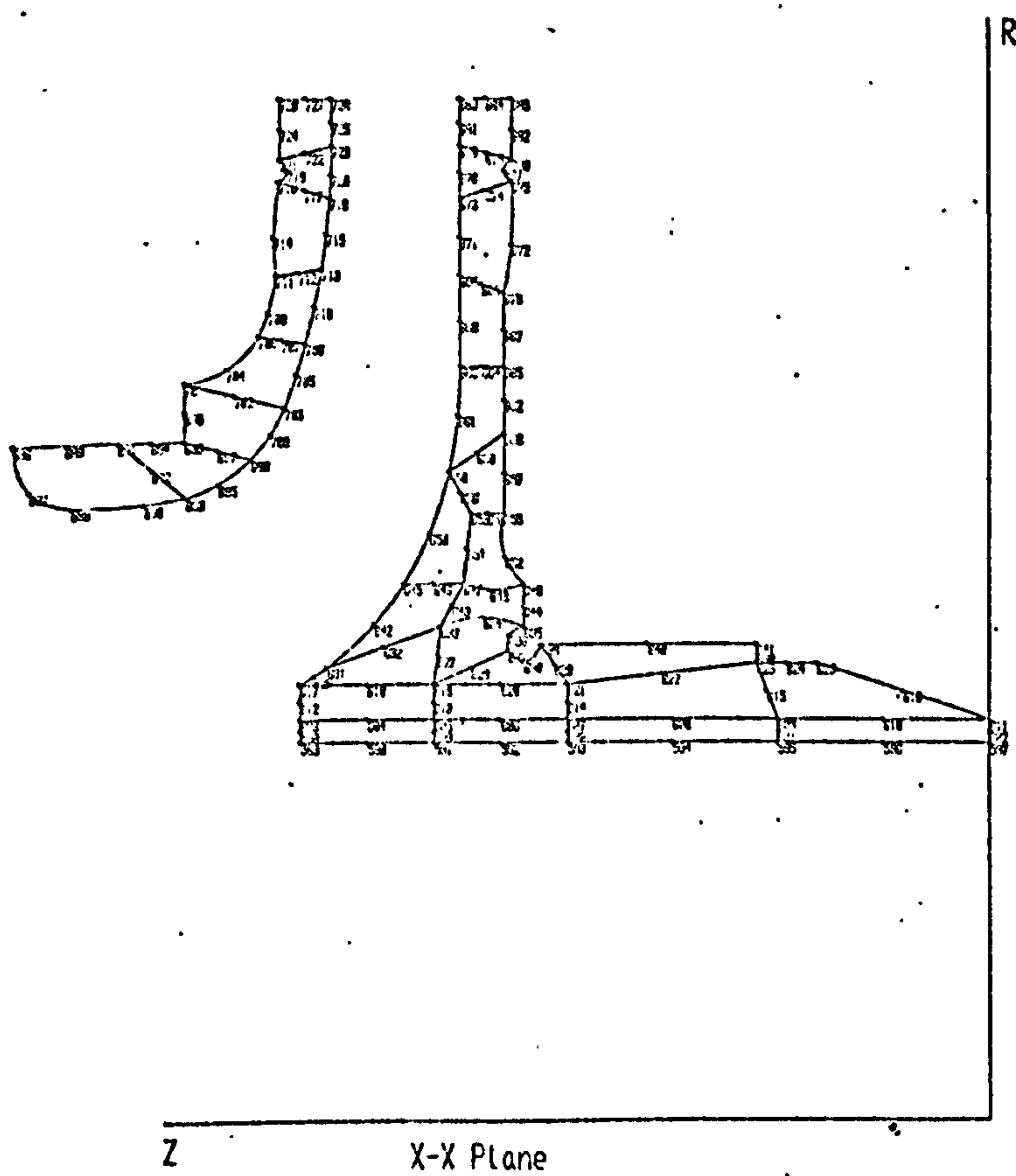
FIG 4-26(c)



ARAMCO TEST IMPELLER

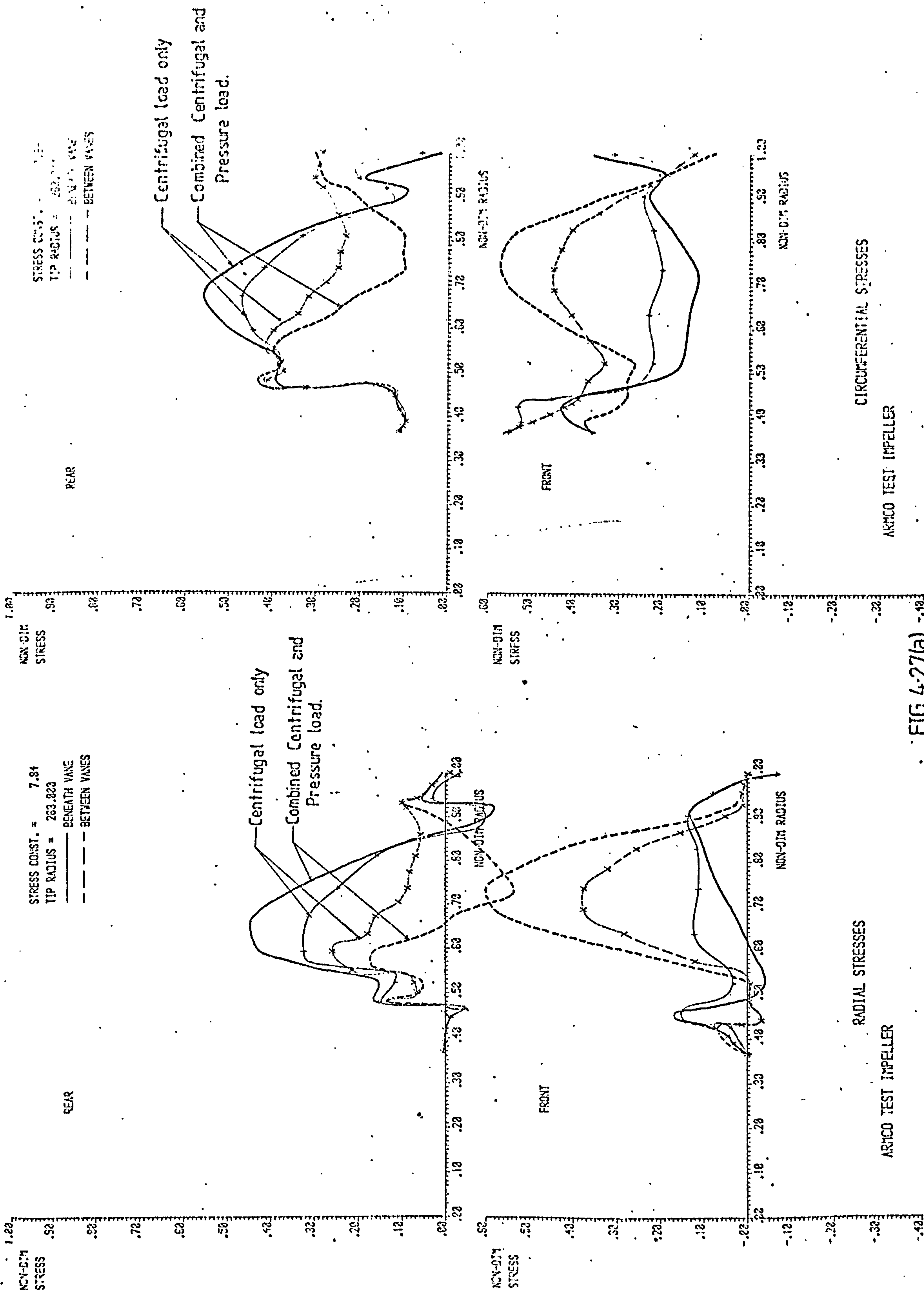


DRAWN IN MM SCALE: .50 TIMES THE SUPPLIED UNITS
Sheet 3



ARAMCO TEST IMPELLER

DRAWN IN MM SCALE: .50 TIMES THE SUPPLIED UNITS
Sheet 4



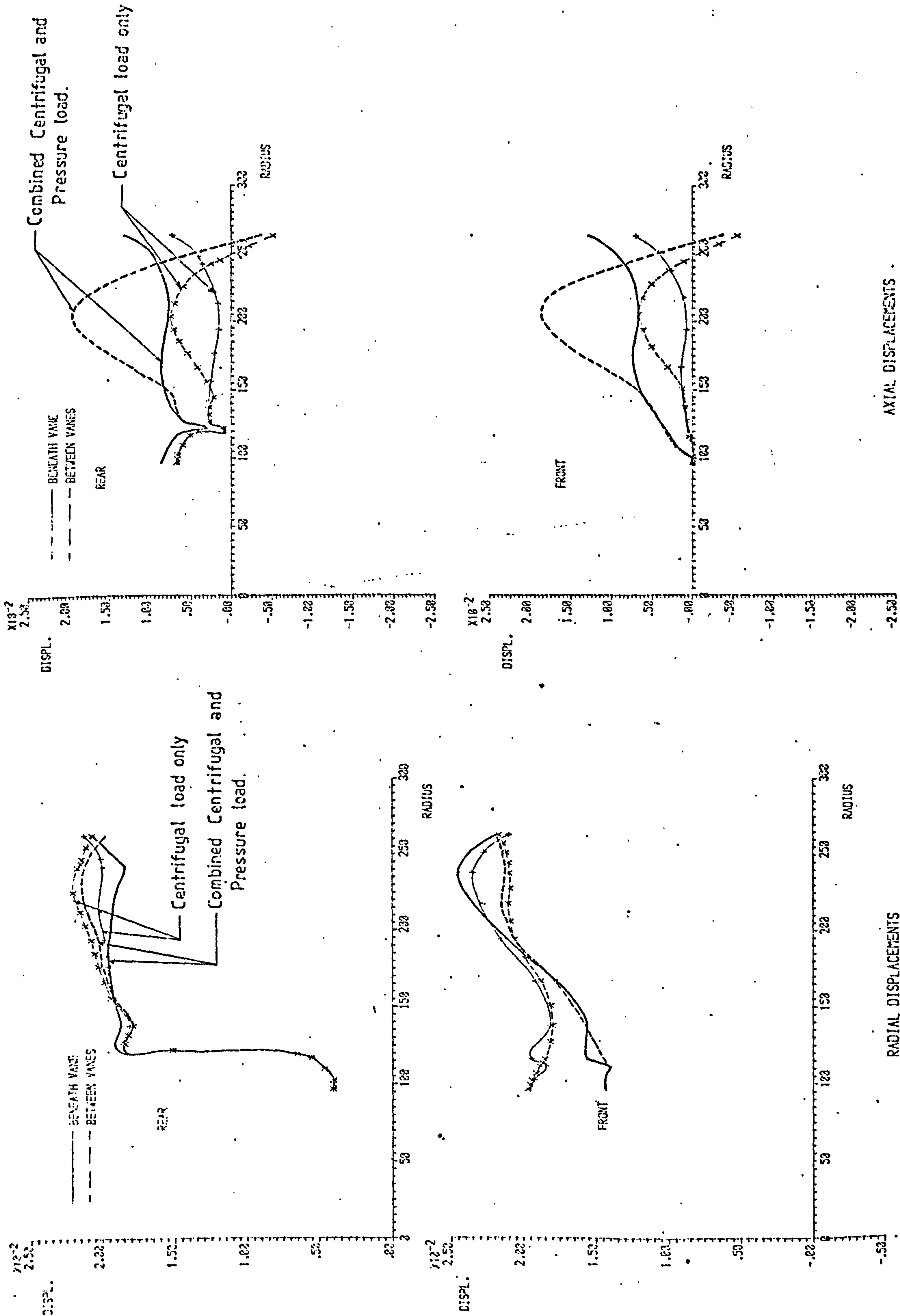


FIG 4-27(b)

AXIAL DISPLACEMENTS
ARMO TEST IMPELLER

RADIAL DISPLACEMENTS
ARMO TEST IMPELLER

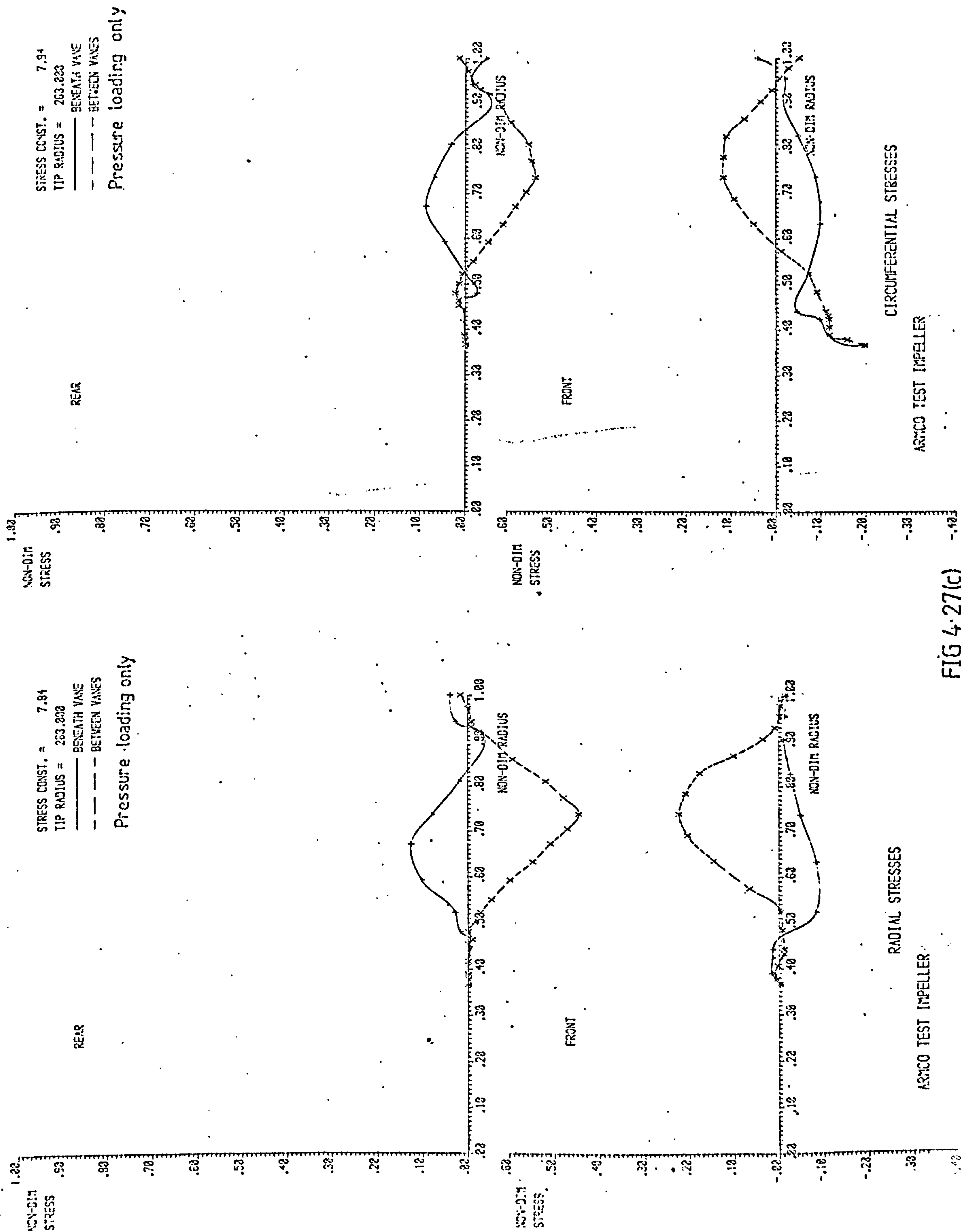
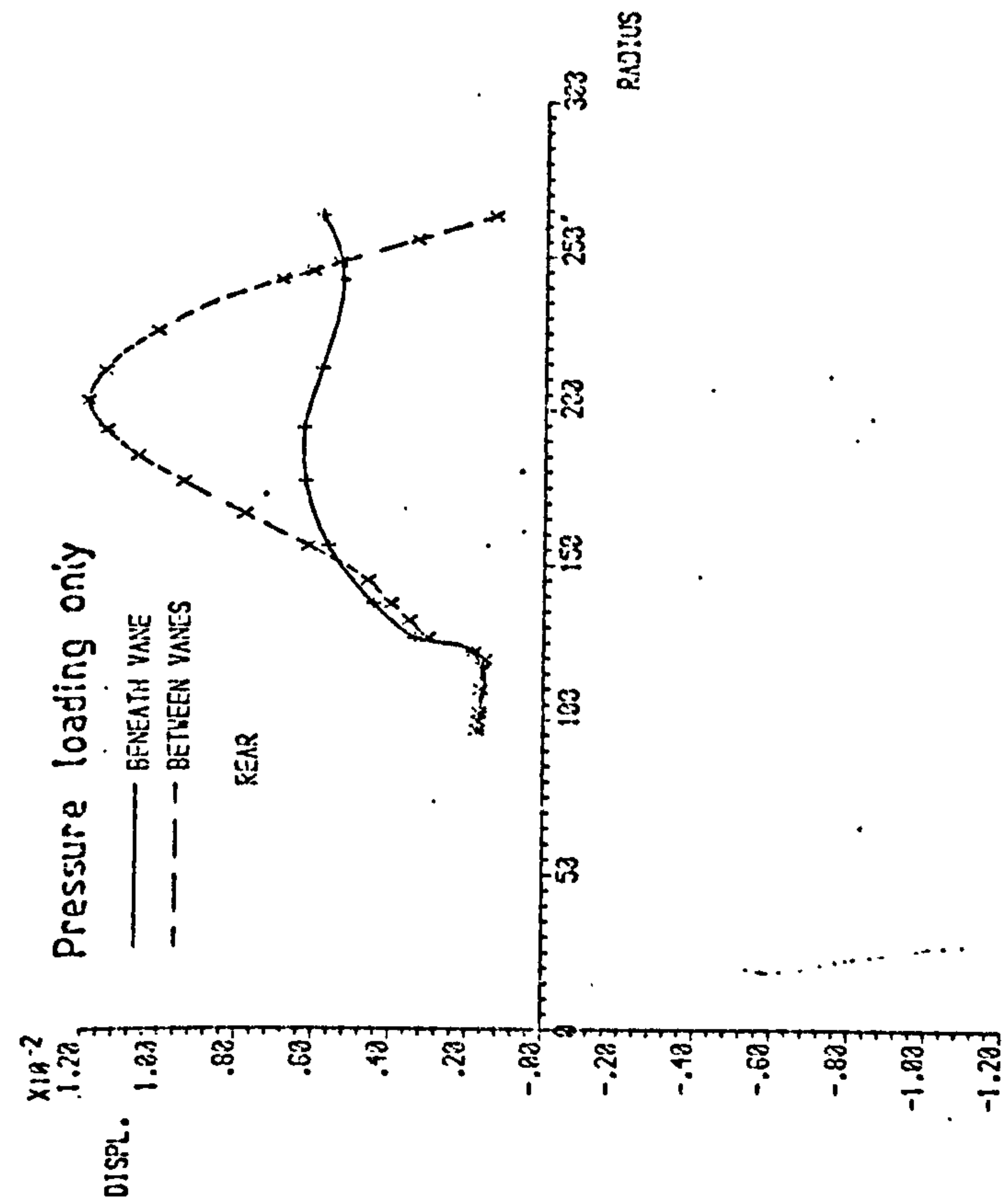
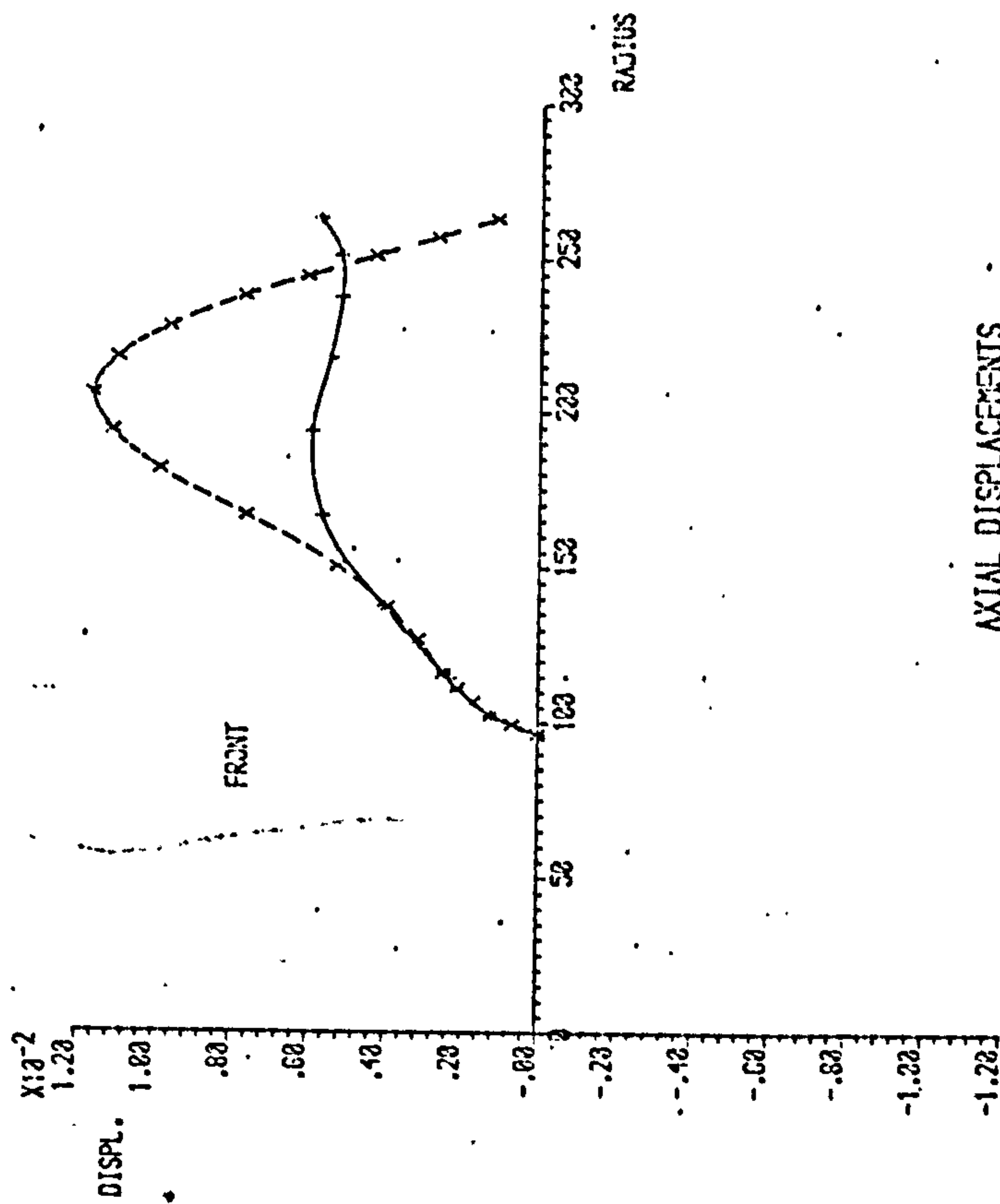
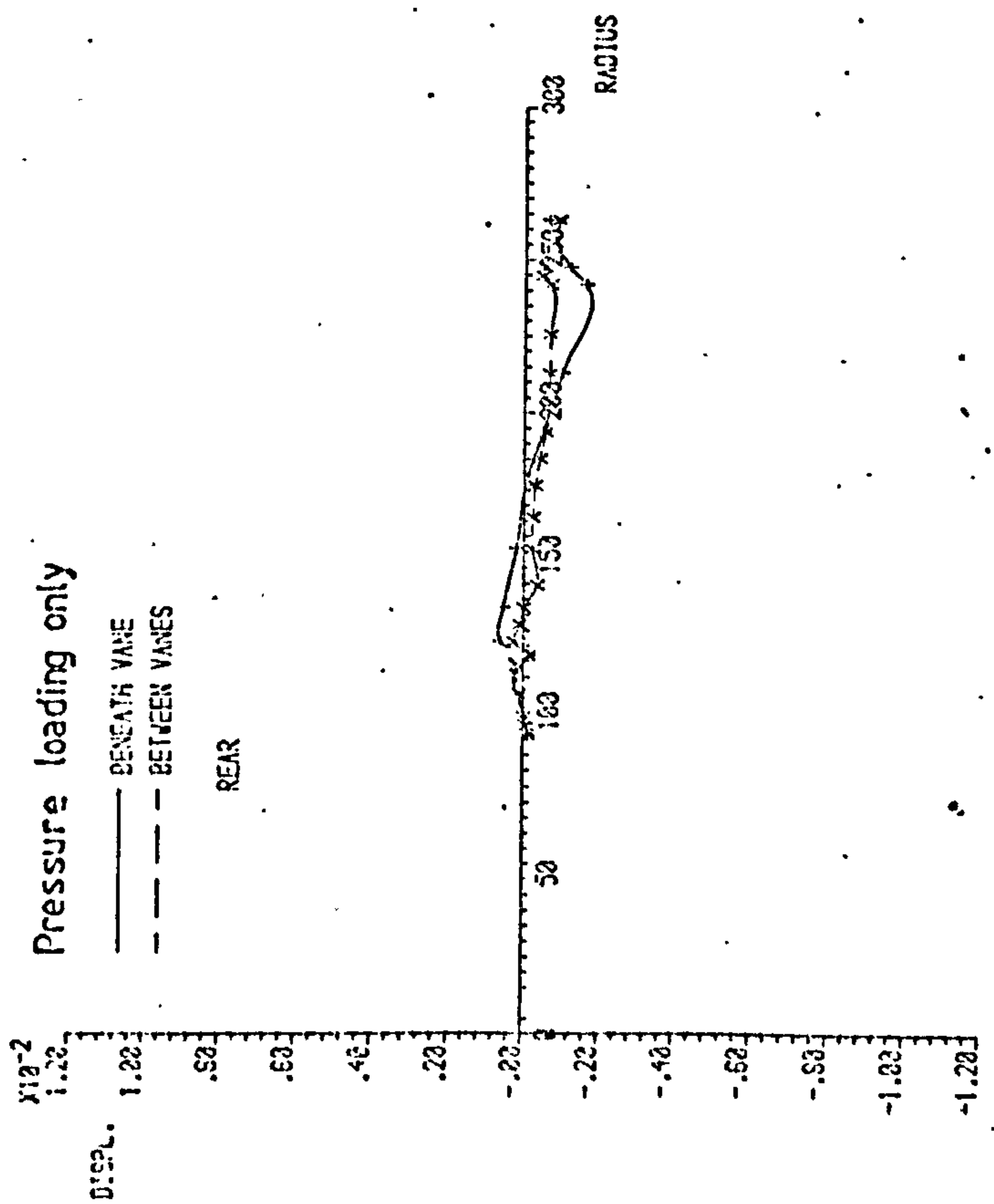
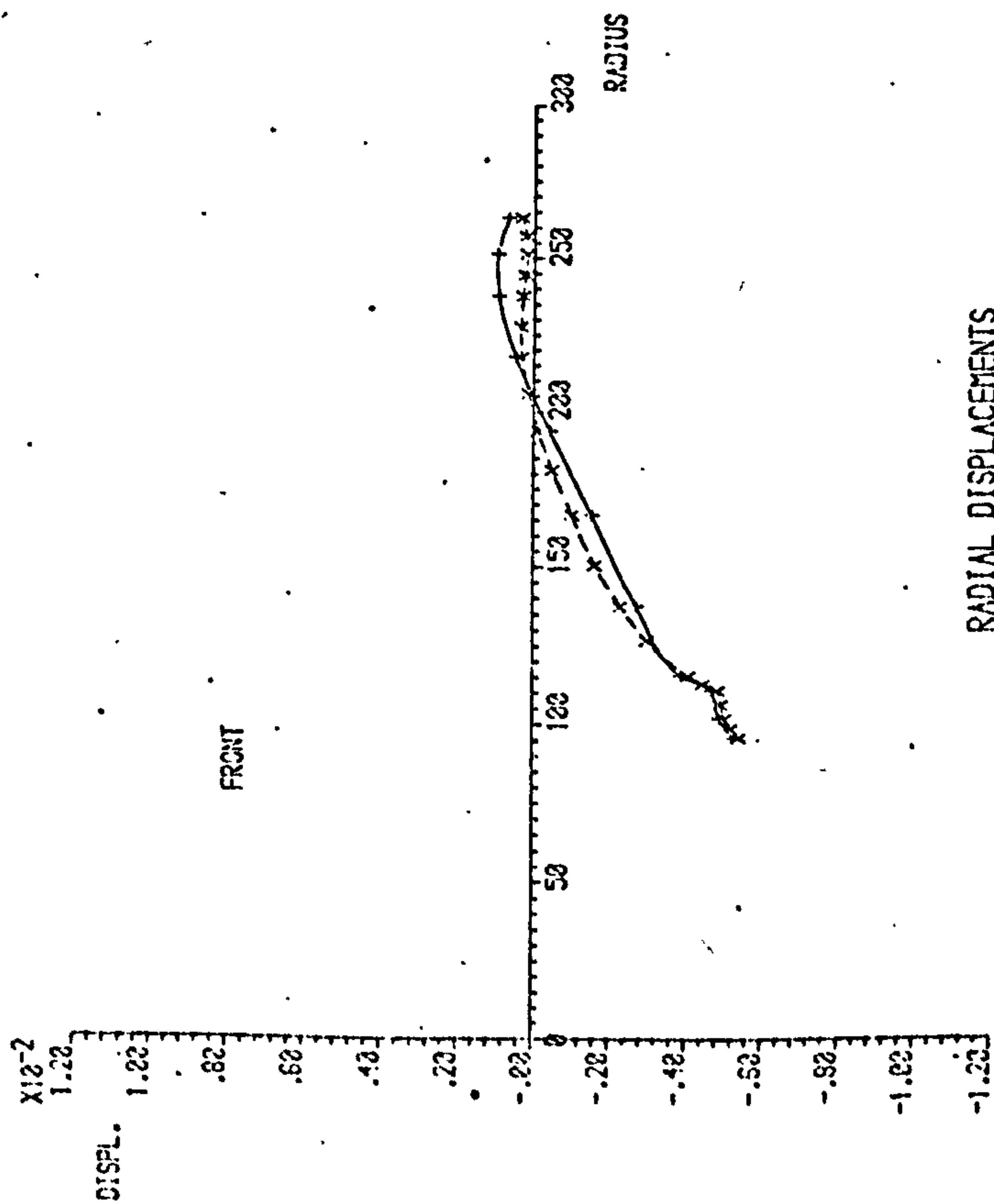


FIG 4-27(c)



AXIAL DISPLACEMENTS
ARMCO TEST IMPELLER

FIG 4-27 (d)

RADIAL DISPLACEMENTS
ARMCO TEST IMPELLER

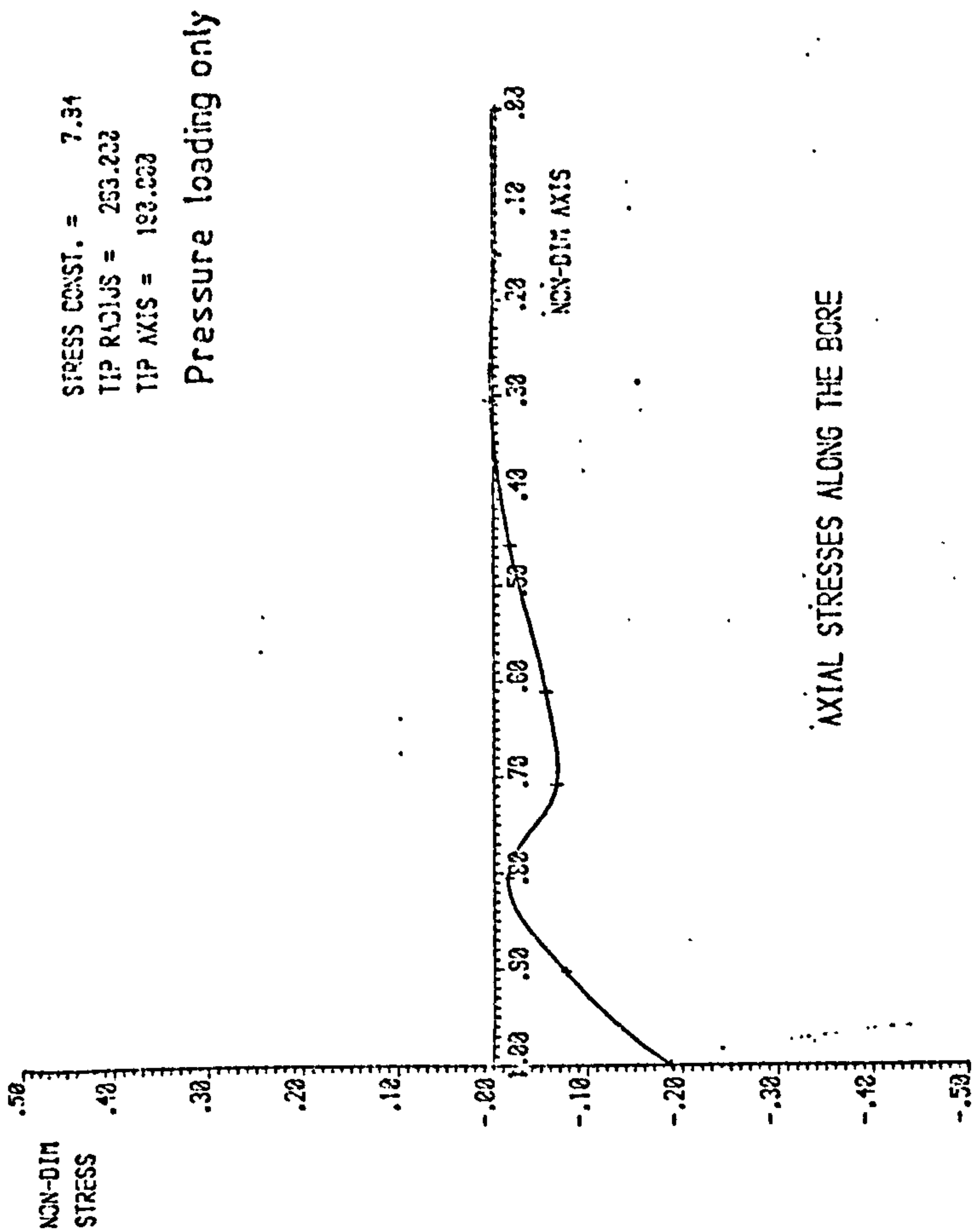
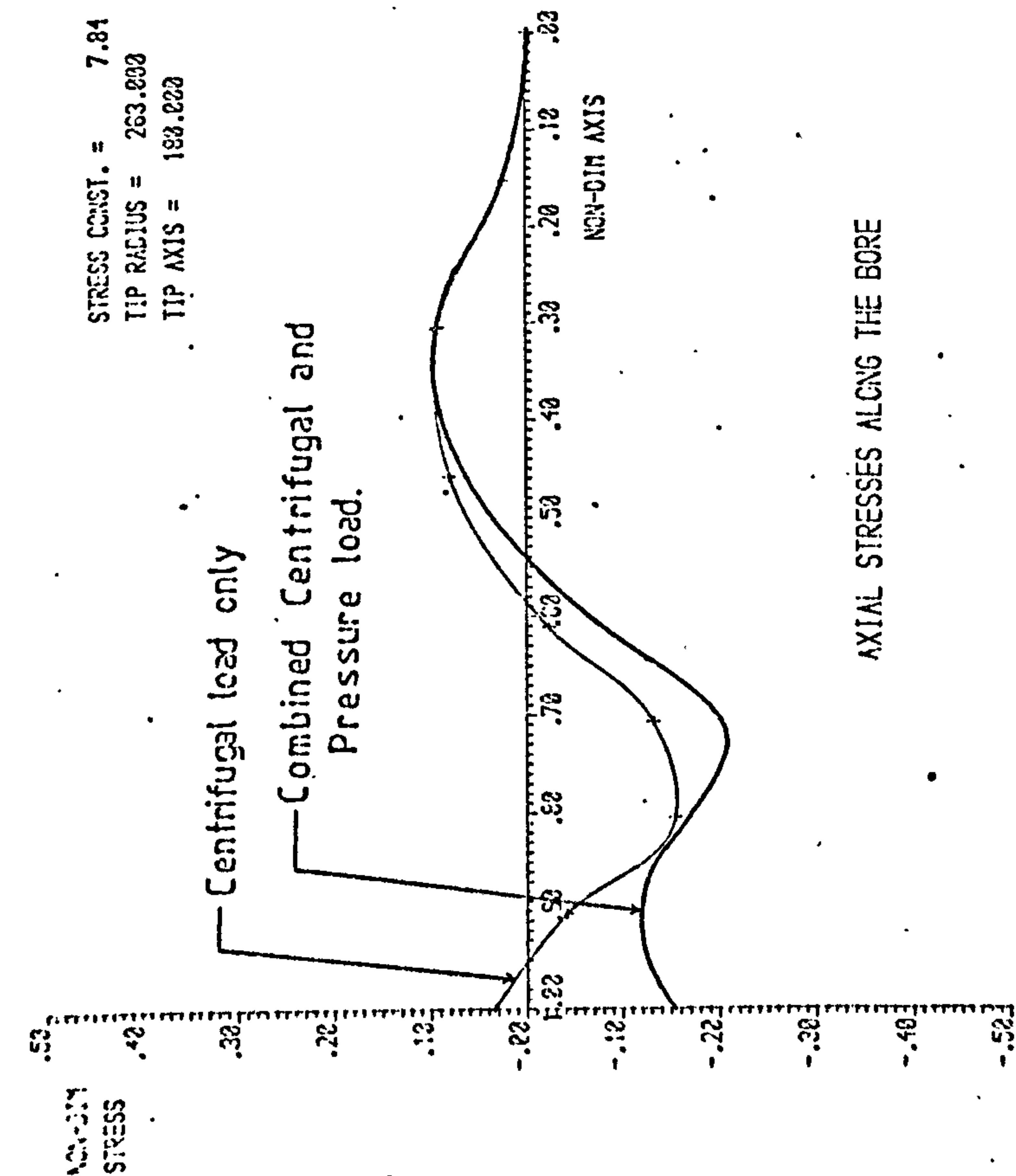
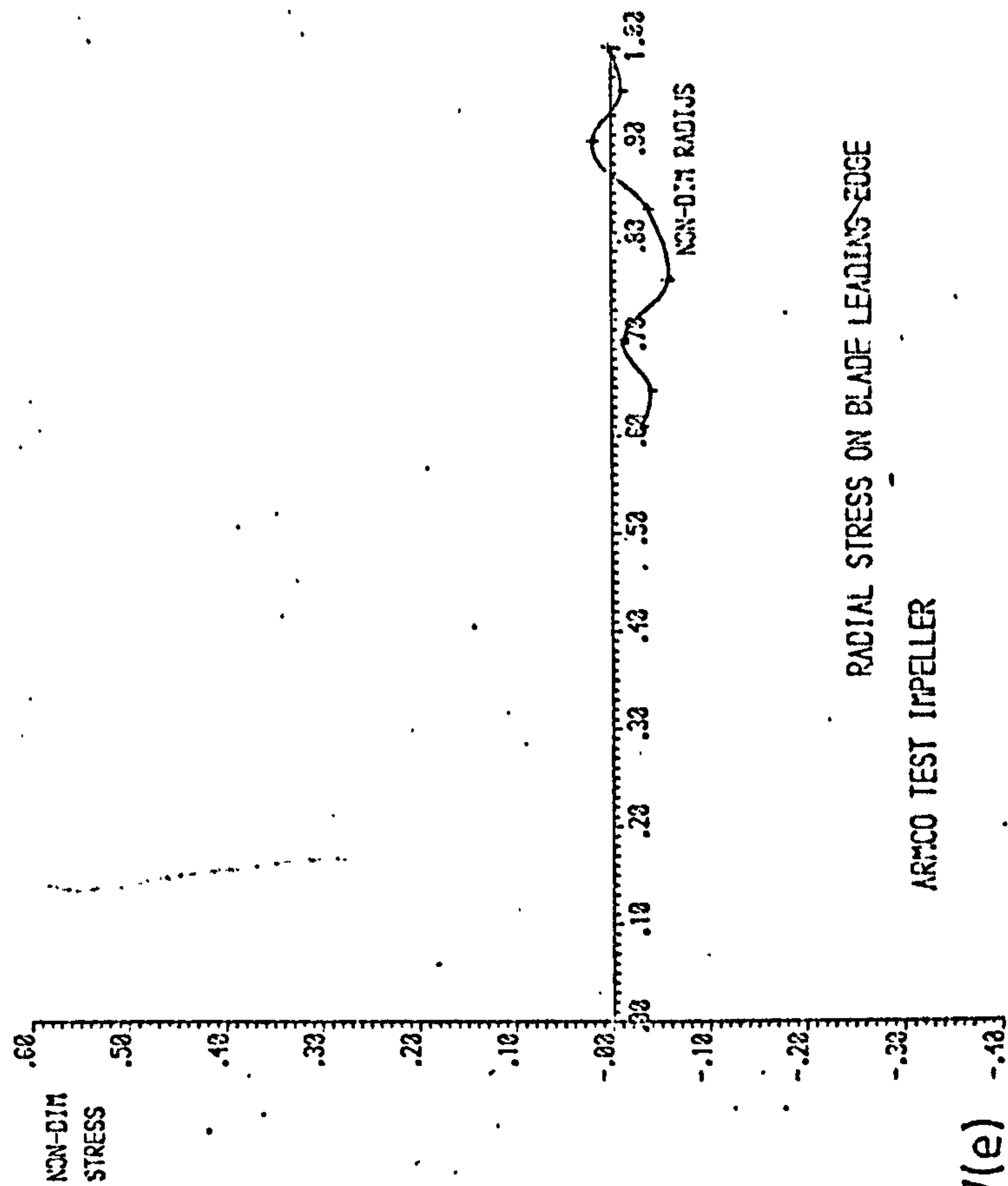
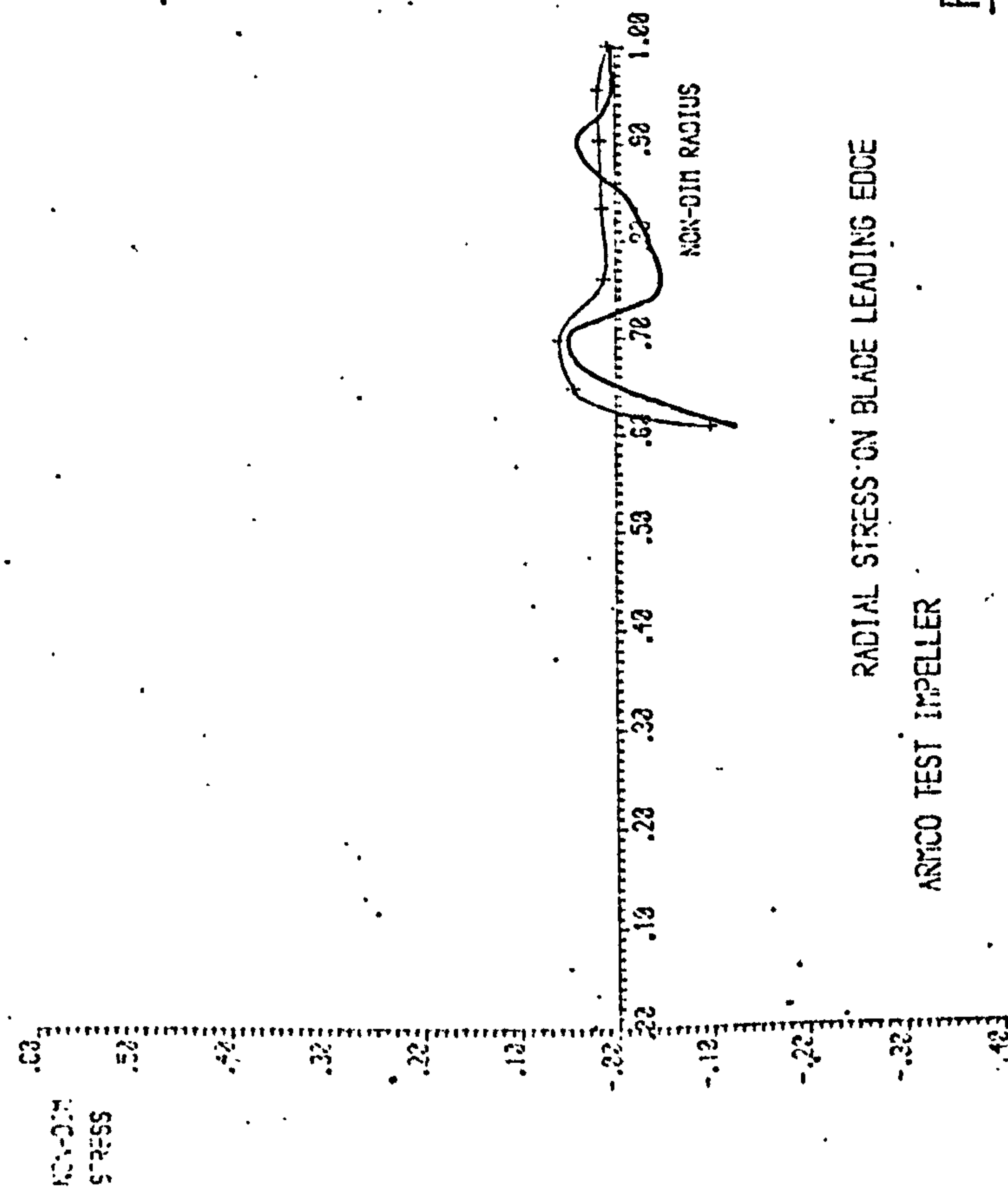


FIG 4-27(e)

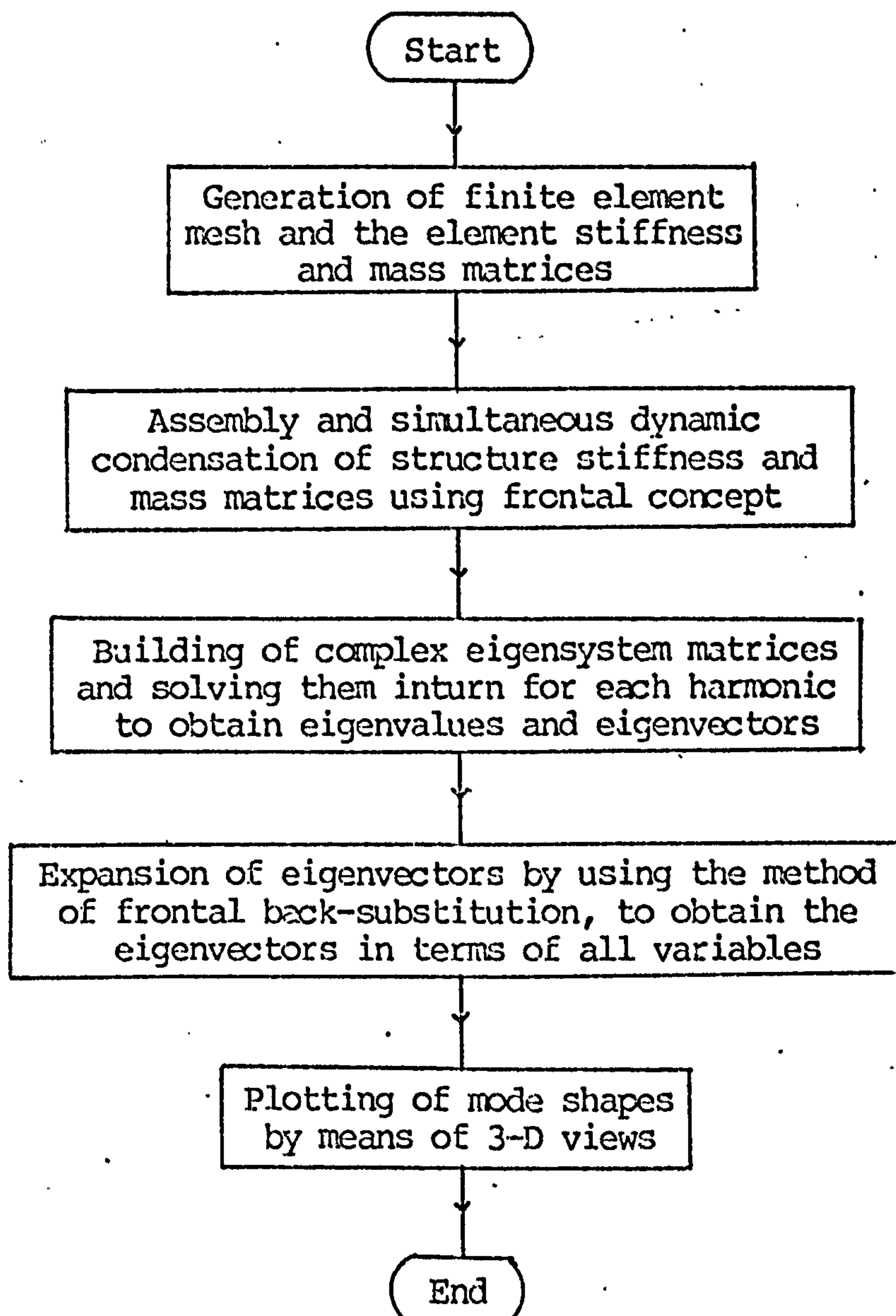


Fig 5.1 Overall flow diagram for dynamic analysis

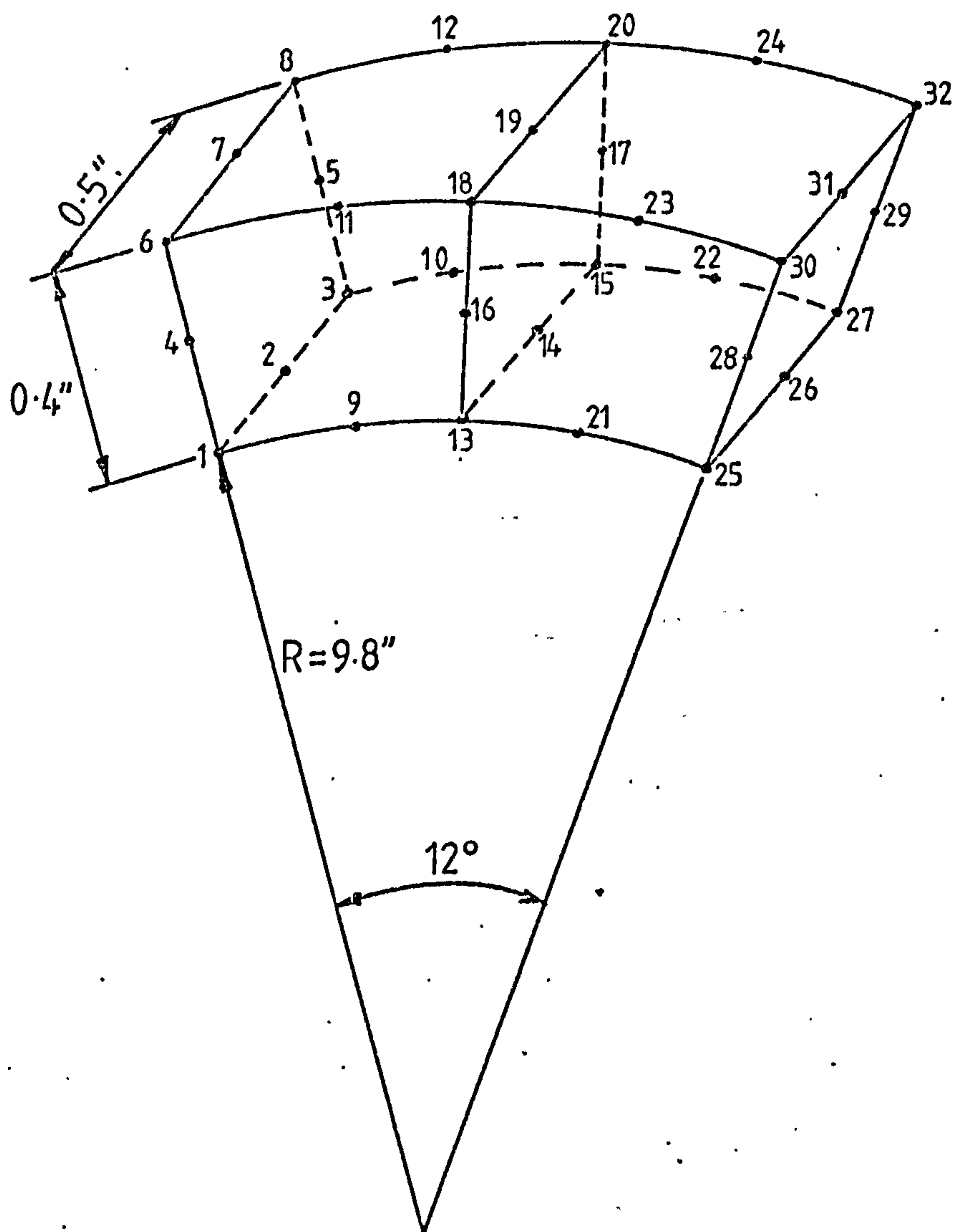
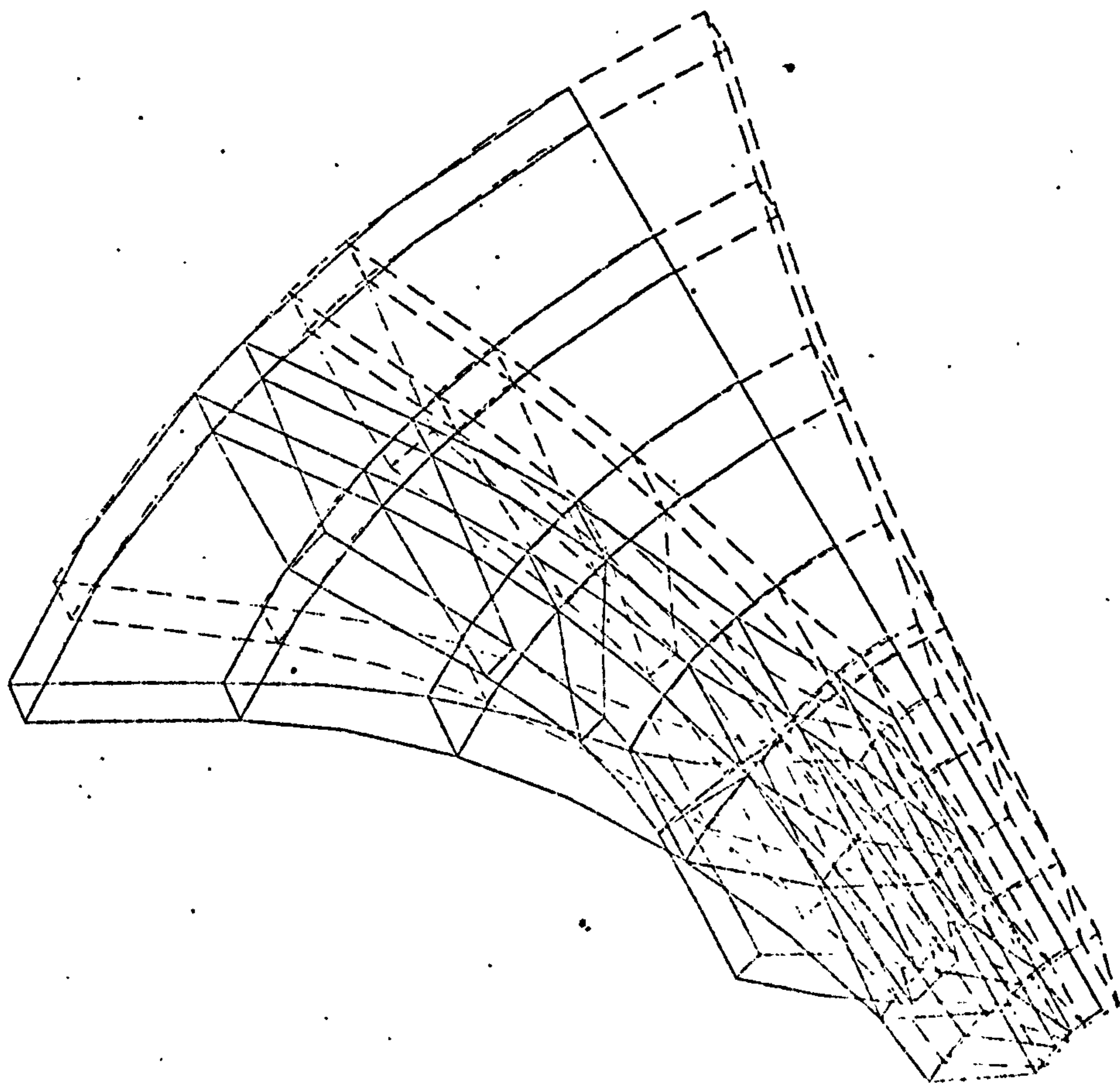


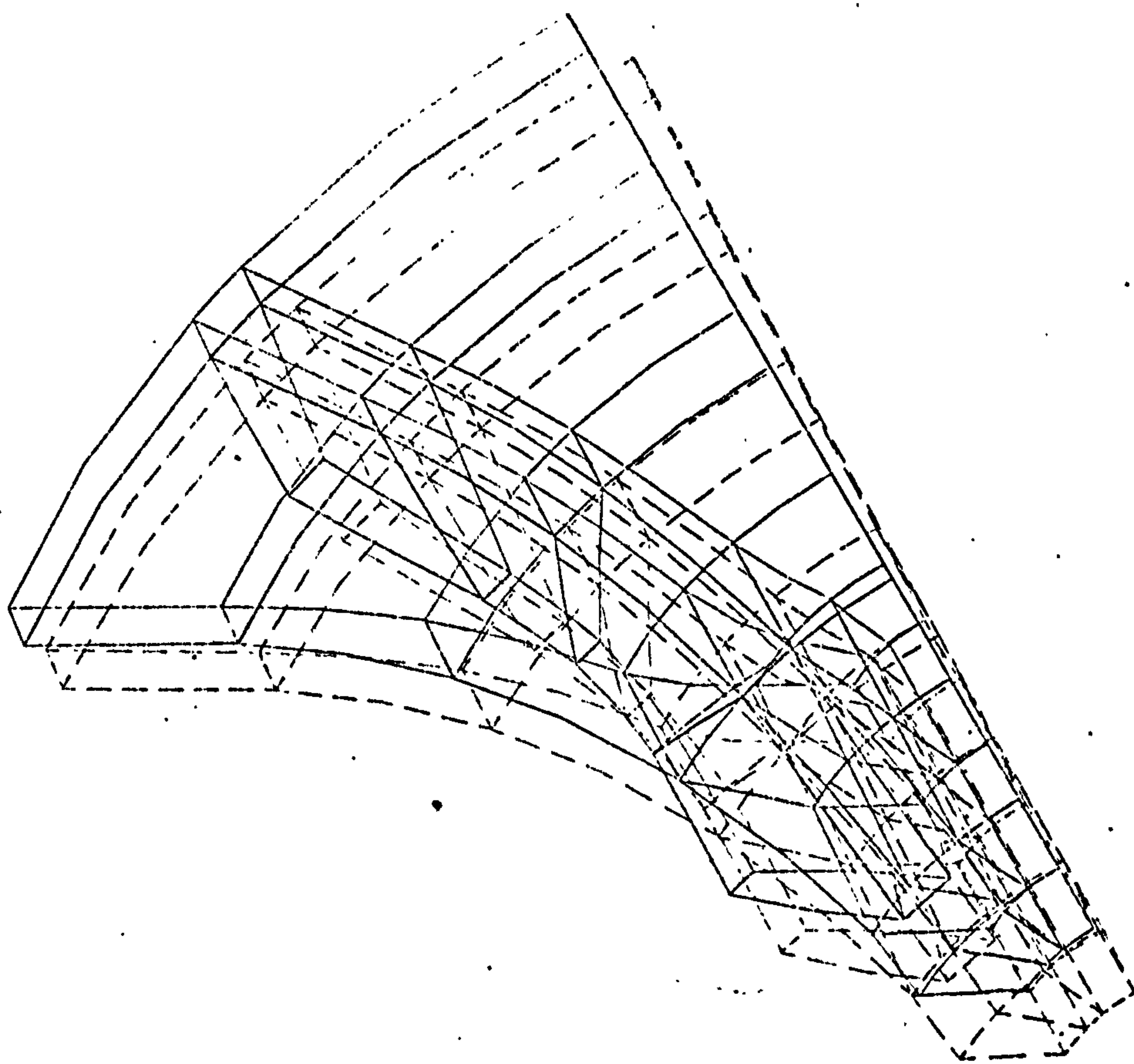
FIG 5.2 A Repeatable Sector of a Ring
Idealised with Two Brick Elements
for Dynamic Analysis.



EIGEN MODE FOR HARMONIC NUMBER 0 FREQUENCY IN CYCLES/SEC = .0

3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN IN SCALE: 55.00 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-P MESH-I 8 BLADES

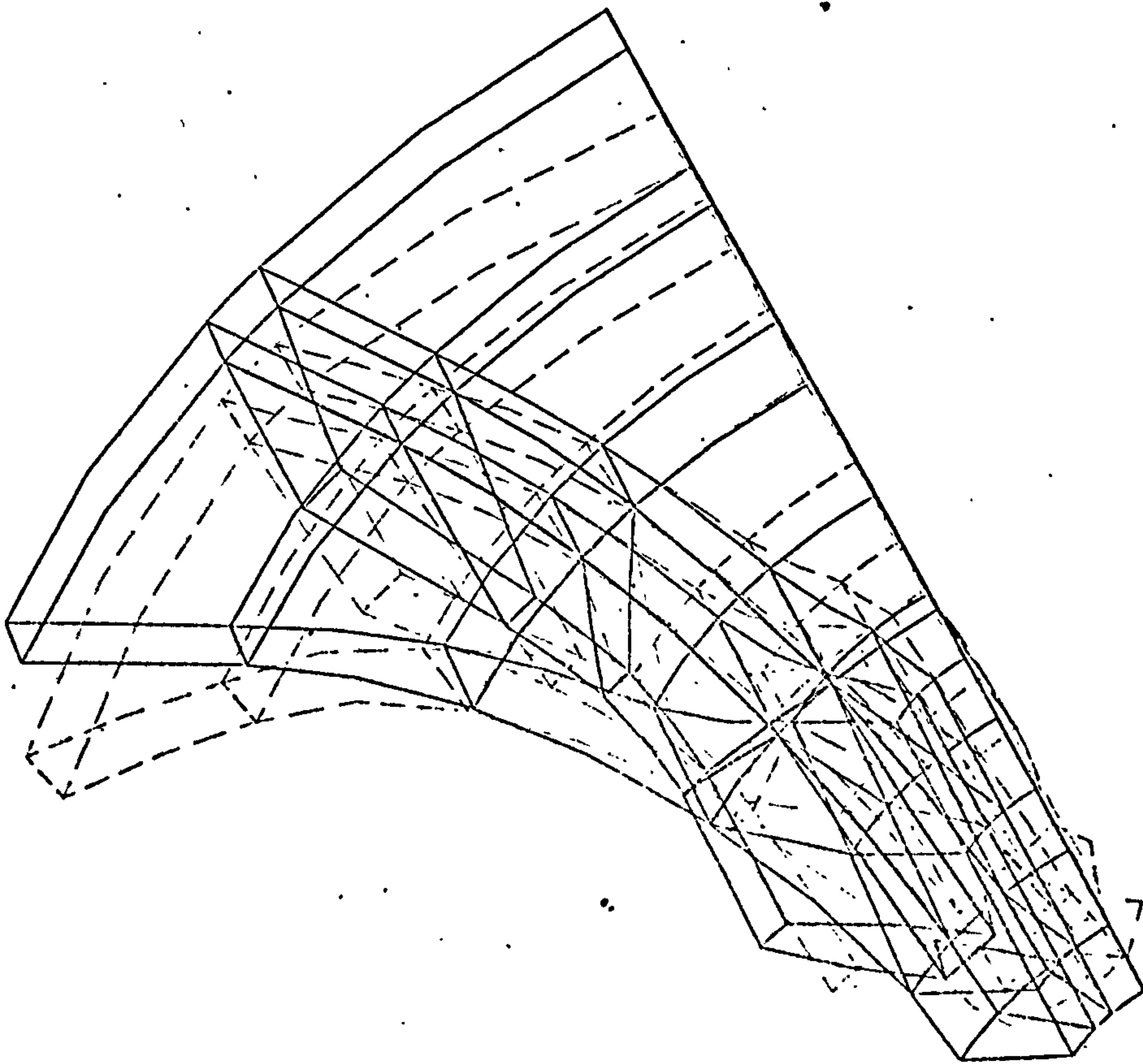


EIGEN MODE FOR HARMONIC NUMBER 0 FREQUENCY IN CYCLES/SEC = .0

3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN IN SCALE: 55.00 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-P MESH-I 8 BLADES

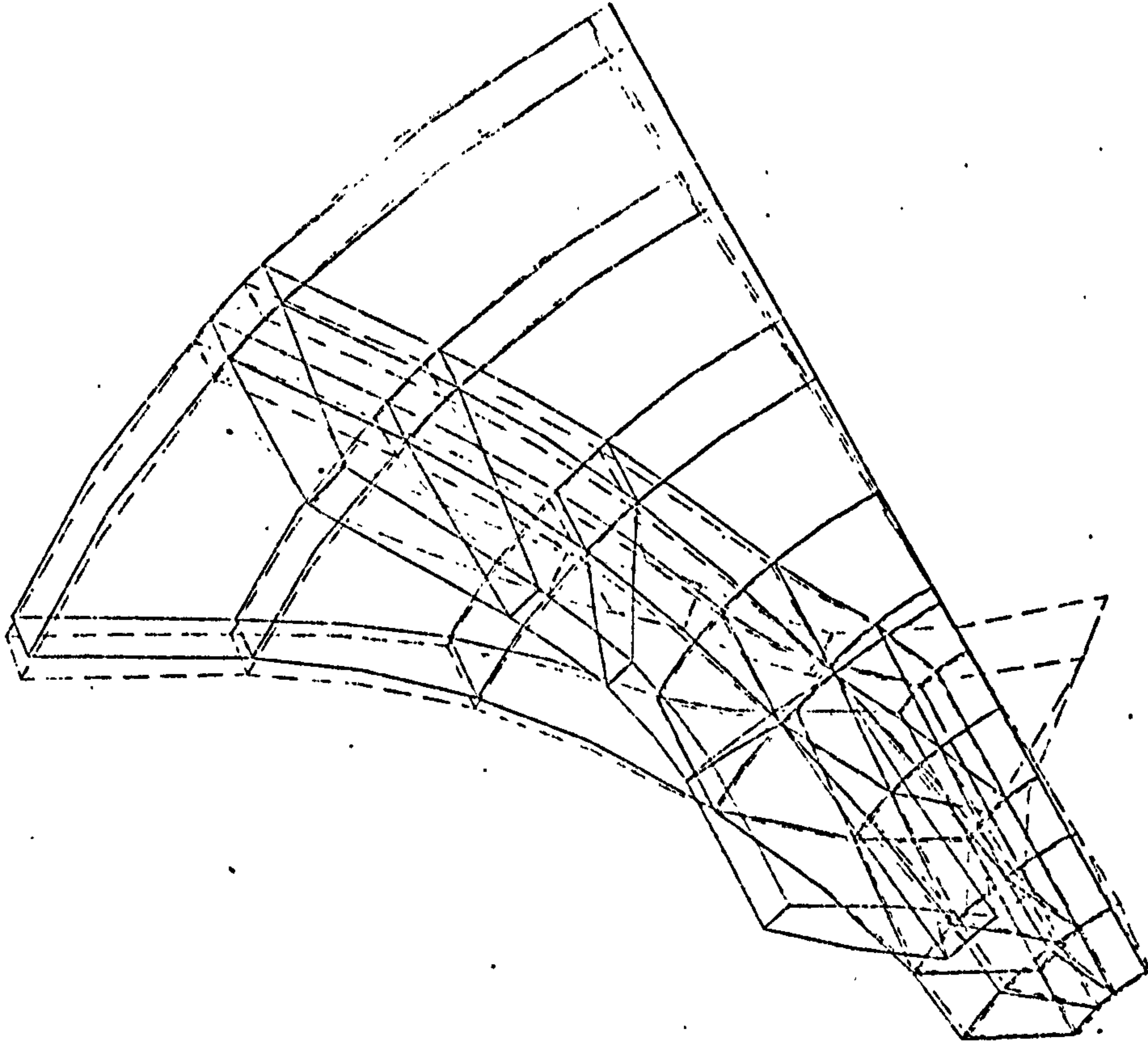
FIG 5-3(a)



EIGEN MODE FOR HARMONIC NUMBER-0 FREQUENCY IN CYCLES/SEC = 4847.3

3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN MM SCALE: 55.23 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-P MESH-I 8 BLADES

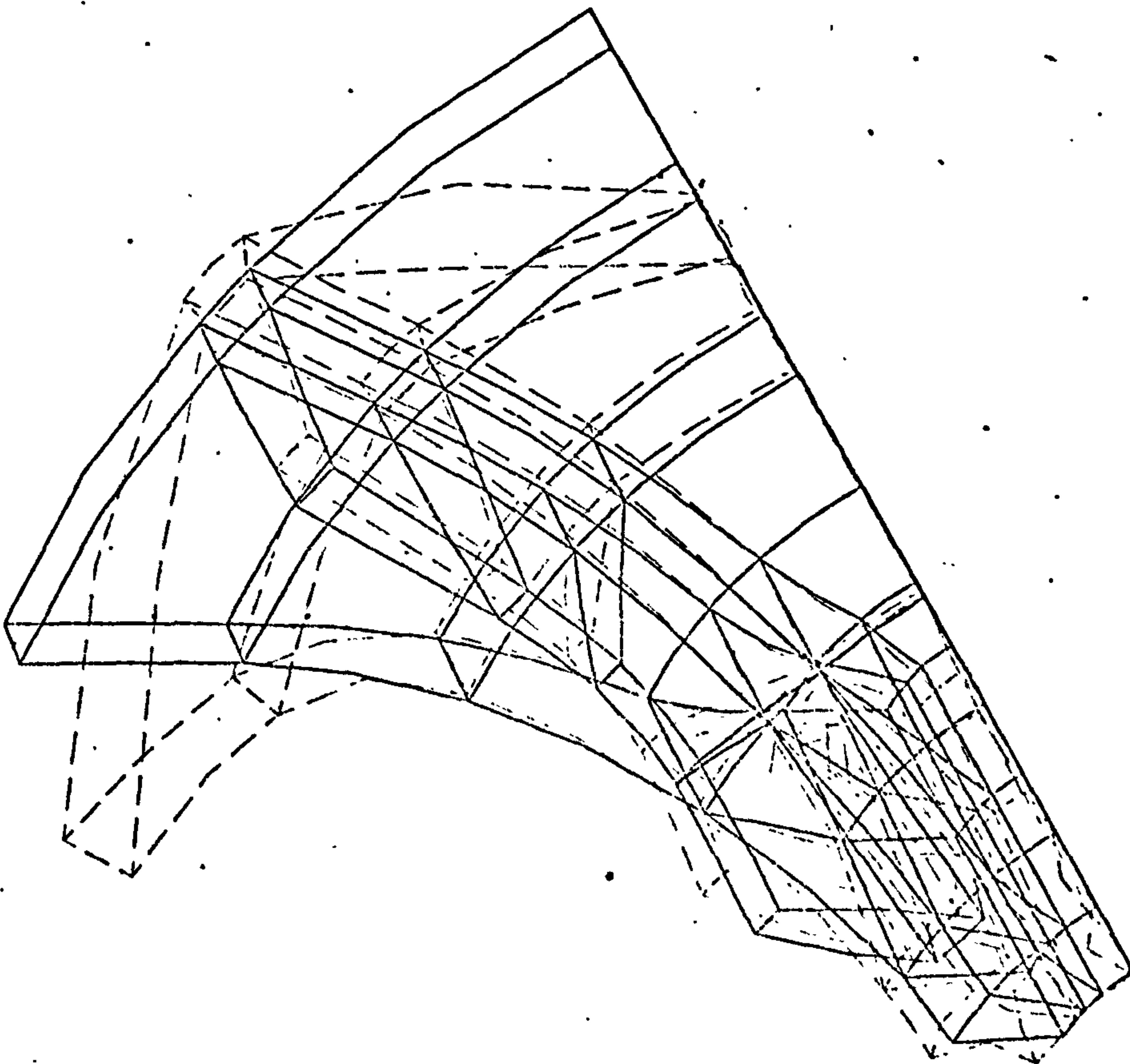


EIGEN MODE FOR HARMONIC NUMBER-0 FREQUENCY IN CYCLES/SEC = 12129.2

3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN MM SCALE: 55.20 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-P MESH-I 8 BLADES

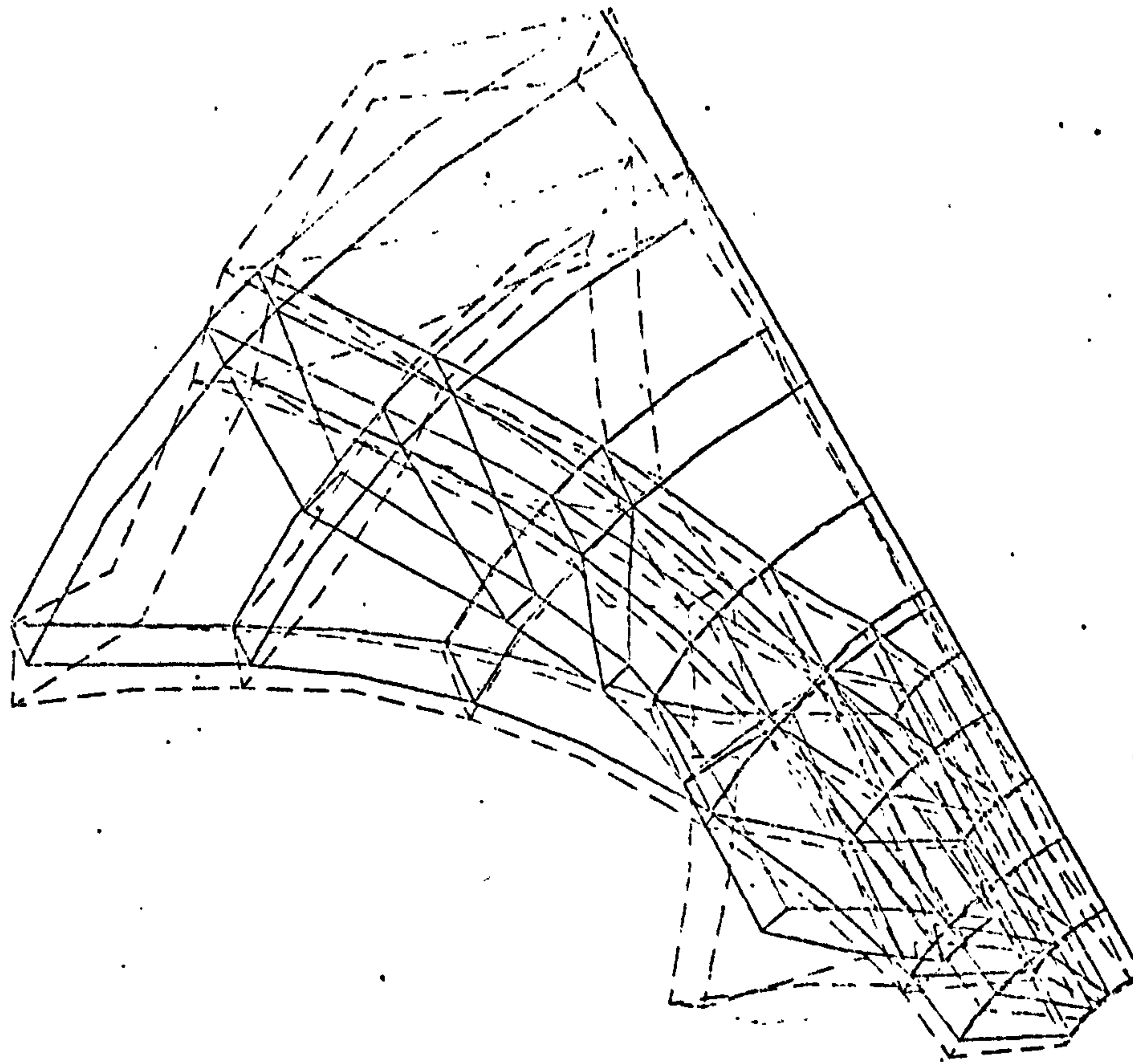
FIG 5-3(b)



EIGEN MODE FOR HARMONIC NUMBER-0 FREQUENCY IN CYCLES/SEC = 11138.4

3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-P MESH-I 8 BLADES

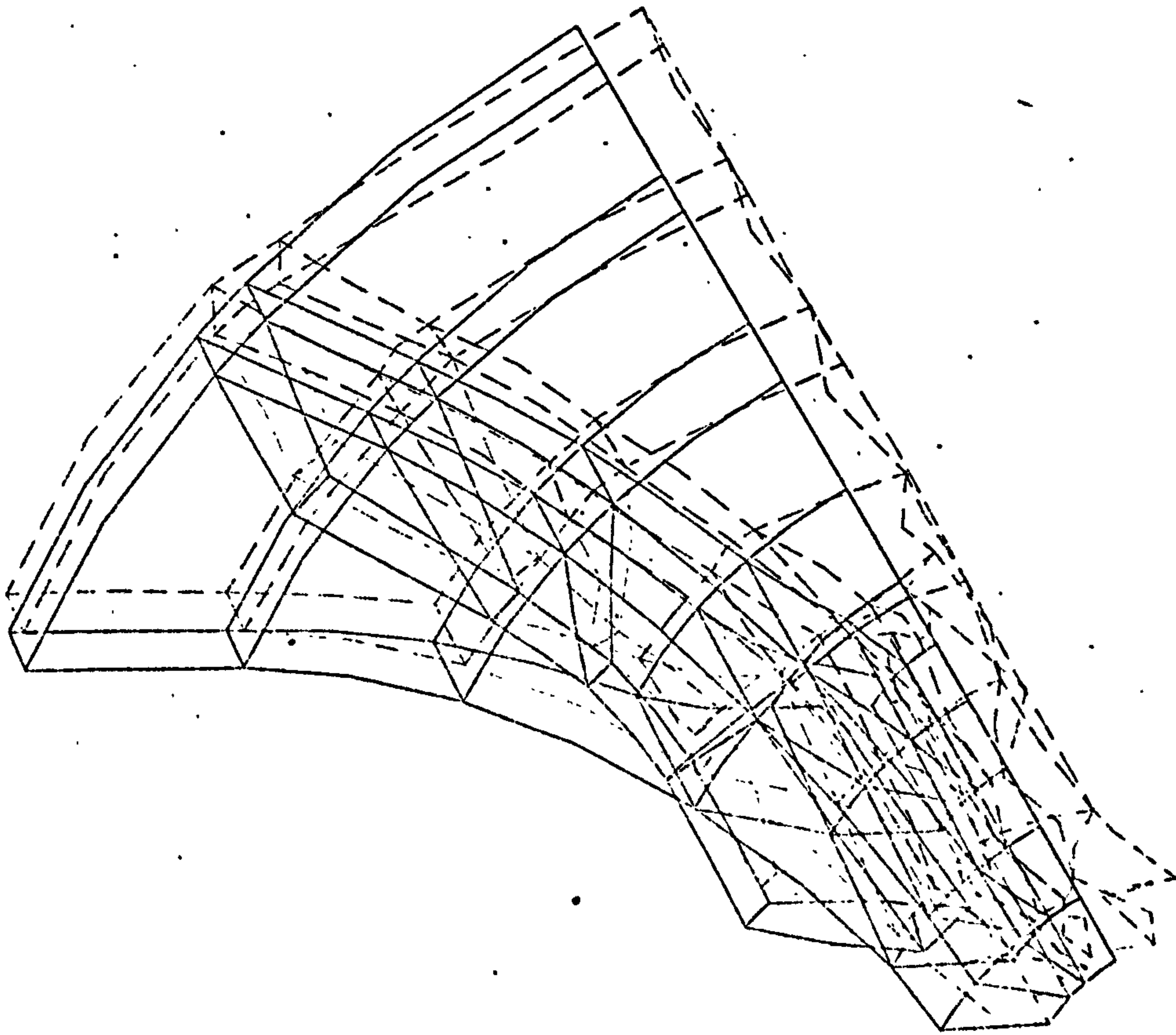


EIGEN MODE FOR HARMONIC NUMBER-0 FREQUENCY IN CYCLES/SEC = 14944.2

3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-P MESH-I 8 BLADES

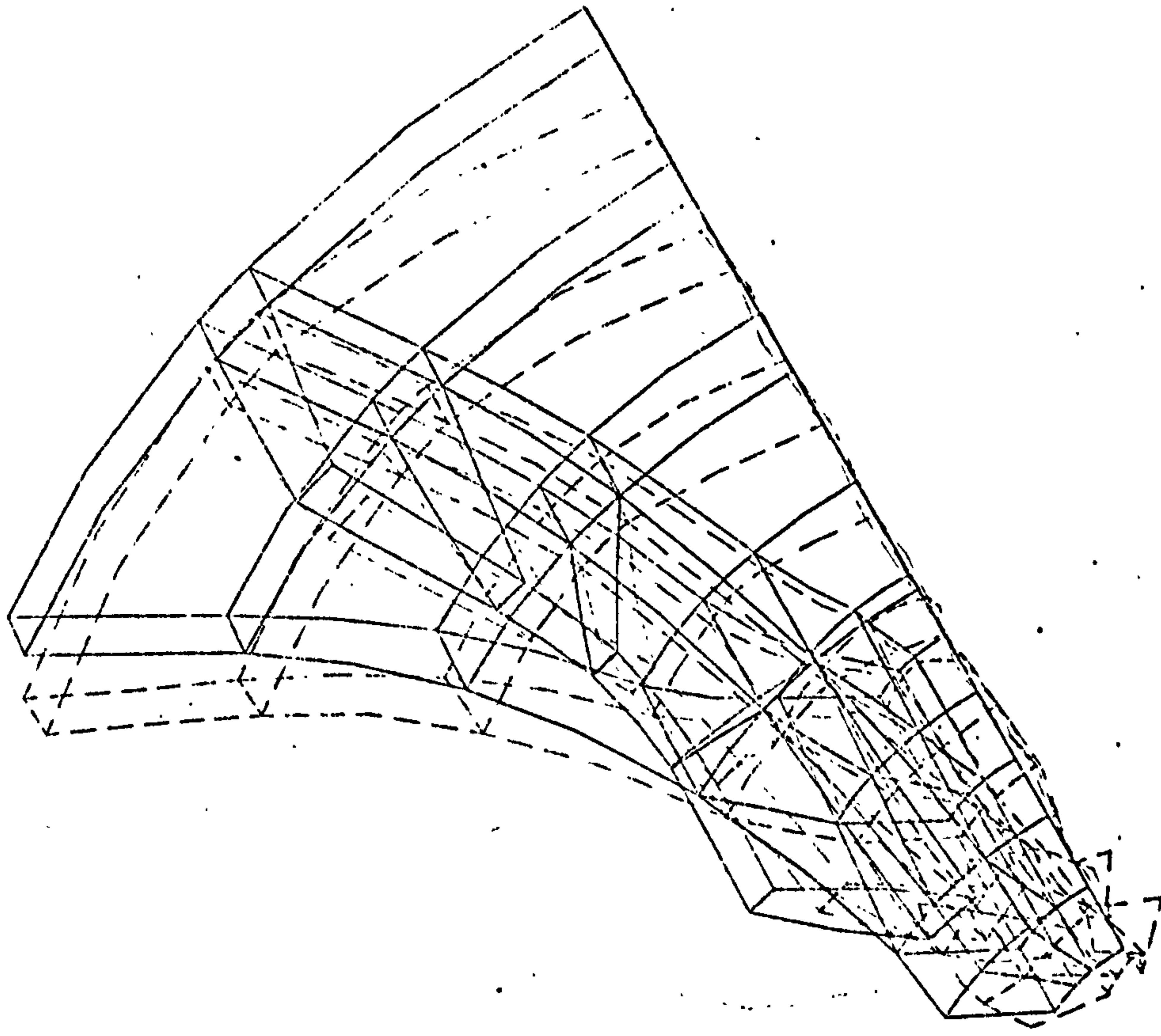
FIG 5.3(c)



EIGEN MODE FOR HARMONIC NUMBER-1 FREQUENCY IN CYCLES/SEC = 26.5

3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-P MESH-I 8 BLADES

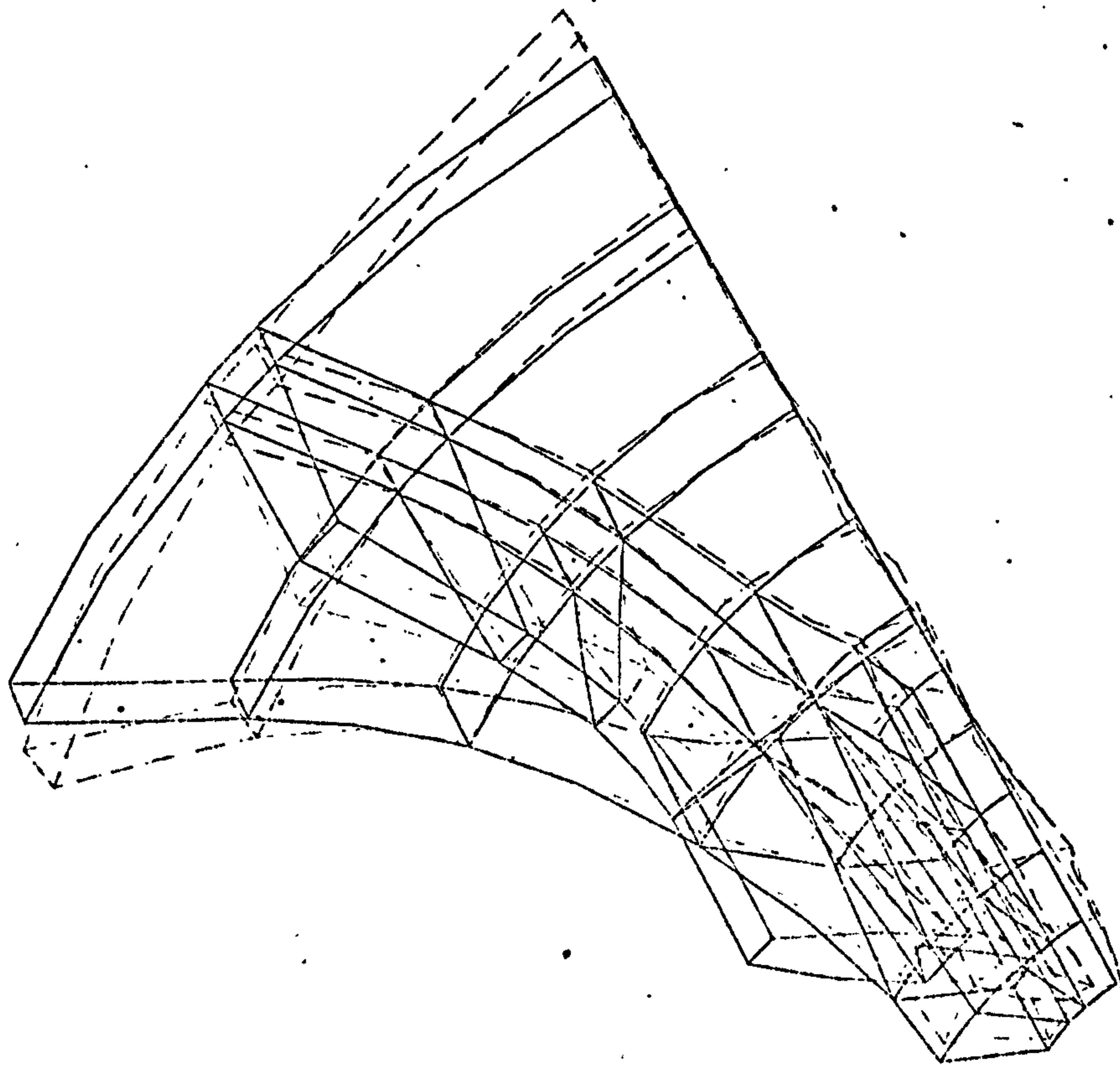


EIGEN MODE FOR HARMONIC NUMBER-1 FREQUENCY IN CYCLES/SEC = 42.1

3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN MM SCALE: 55.00 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-P MESH-I 8 BLADES

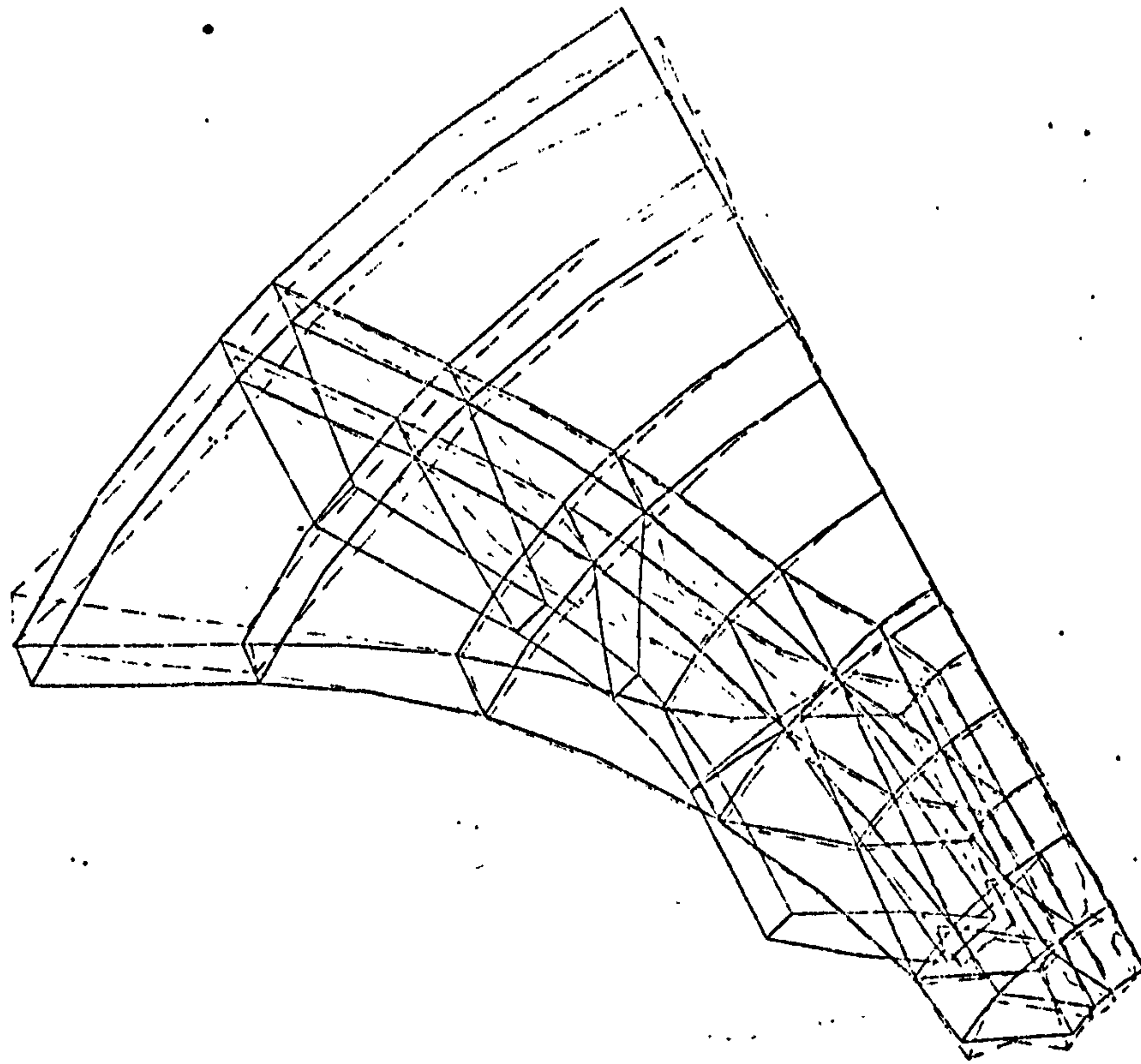
FIG 5-3 (d)



EIGEN MODE FOR HARMONIC NUMBER-1 FREQUENCY IN CYCLES/SEC = 8588.6

3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN IN SCALE: 55.80 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-P MESH-1 8 BLADES

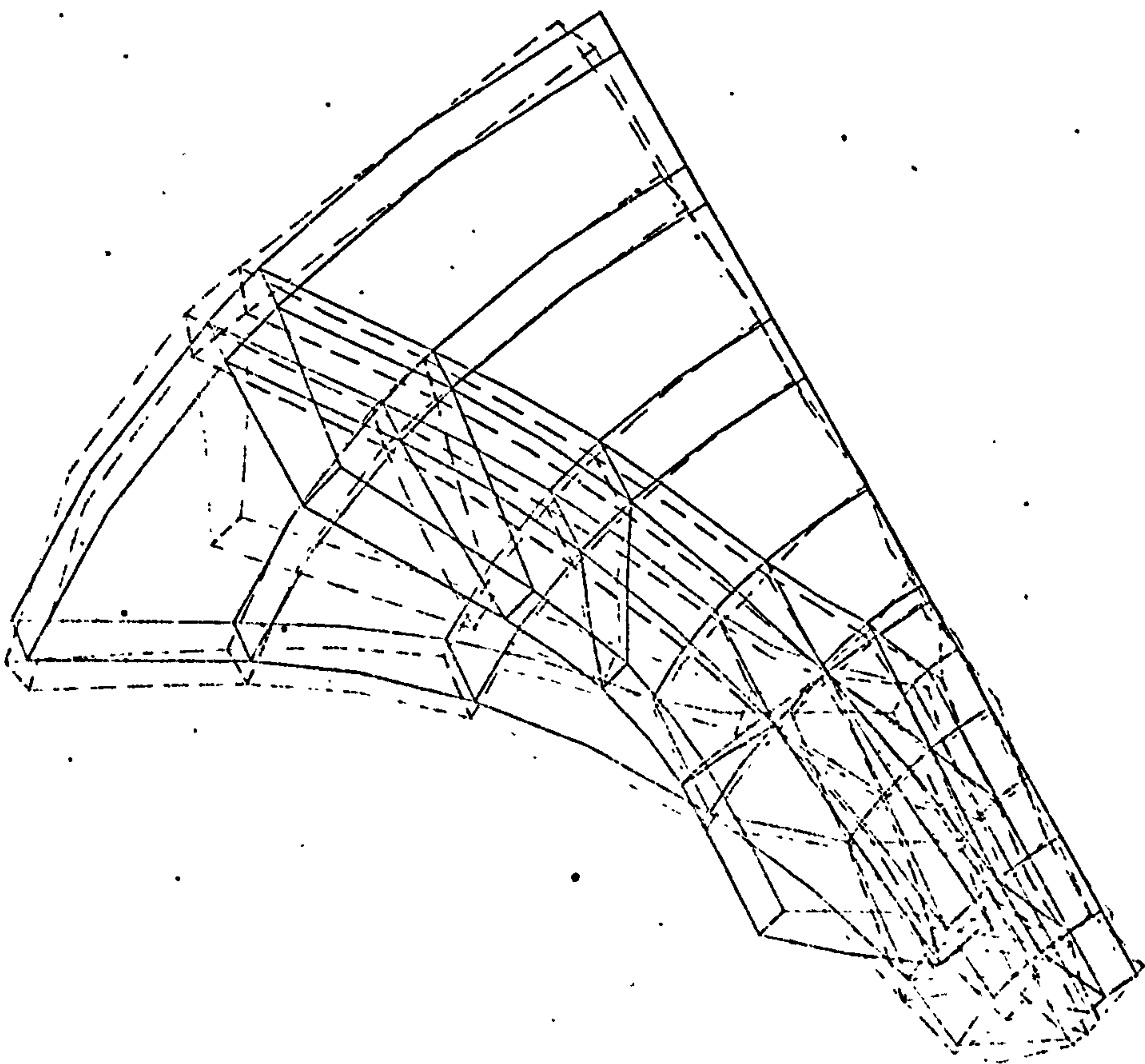


EIGEN MODE FOR HARMONIC NUMBER-1 FREQUENCY IN CYCLES/SEC = 9544.7

3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN IN SCALE: 55.83 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-P MESH-1 8 BLADES

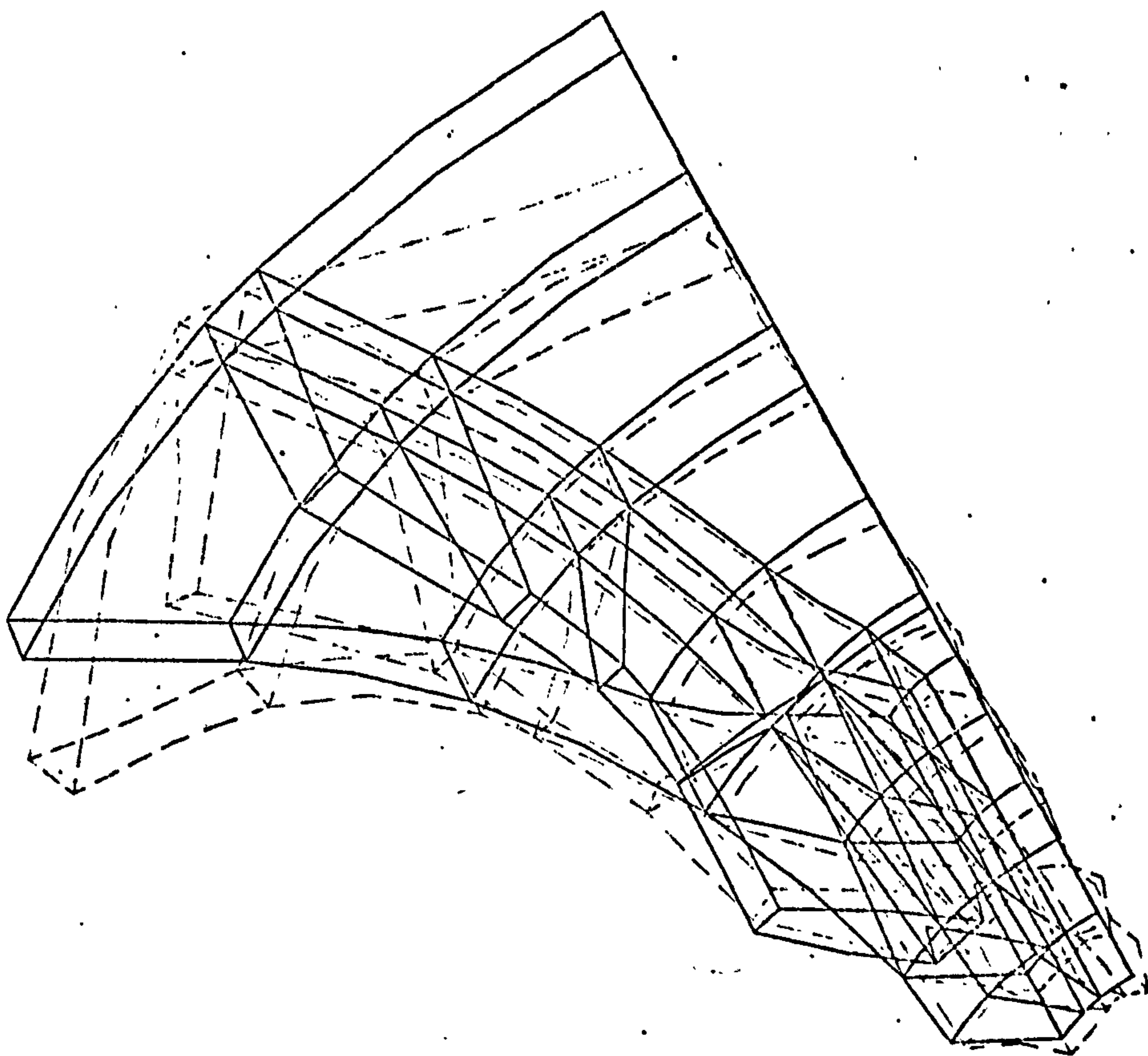
FIG 5-3 (e)



EIGEN MODE FOR HARMONIC NUMBER-1 FREQUENCY IN CYCLES/SEC = 12577.7

3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN FT SCALE: 55.00 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-P MESH-1 8 BLADES

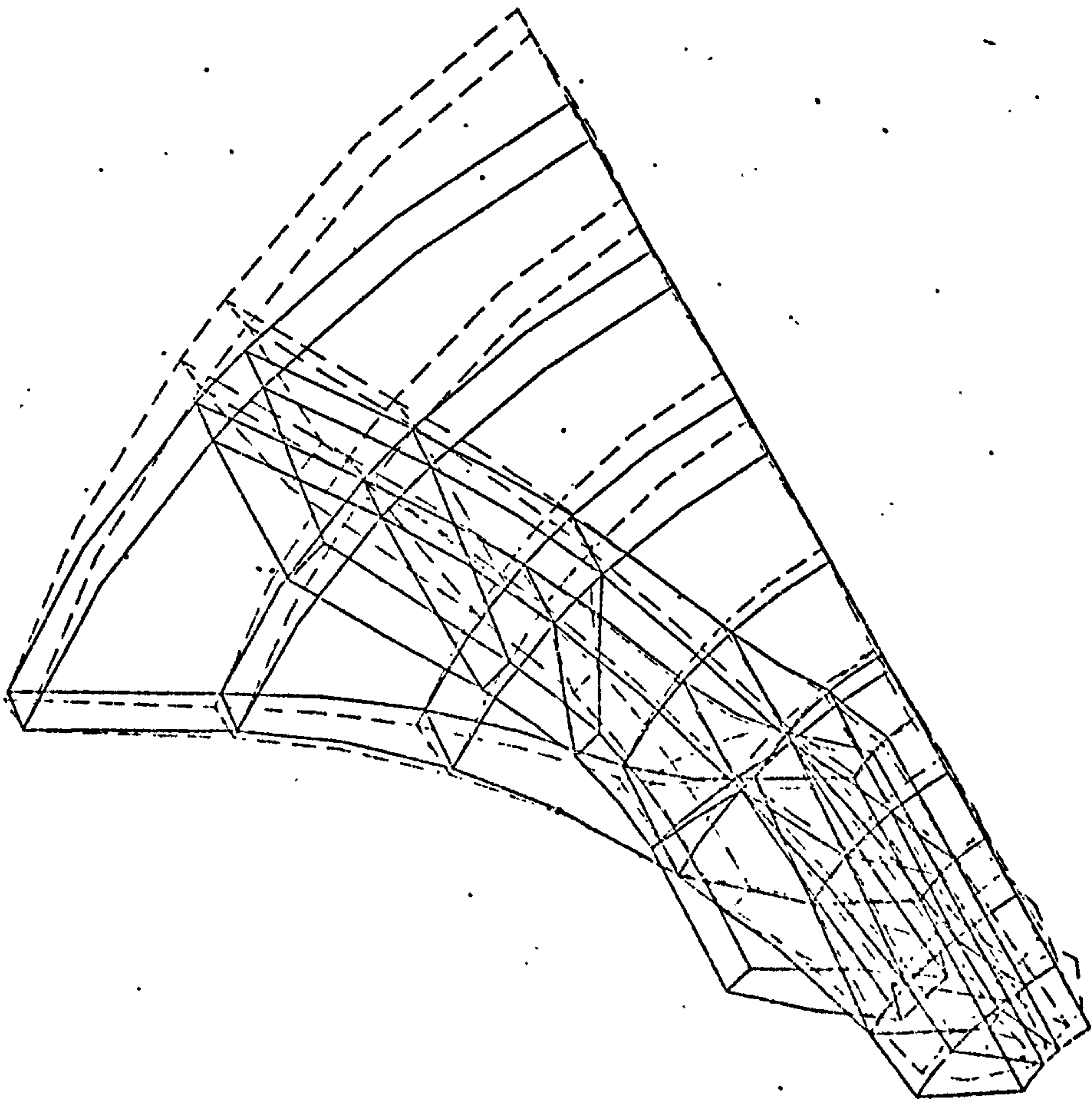


EIGEN MODE FOR HARMONIC NUMBER-1 FREQUENCY IN CYCLES/SEC = 14233.5

3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN FT SCALE: 55.00 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-P MESH-1 8 BLADES

FIG 5-3(f)

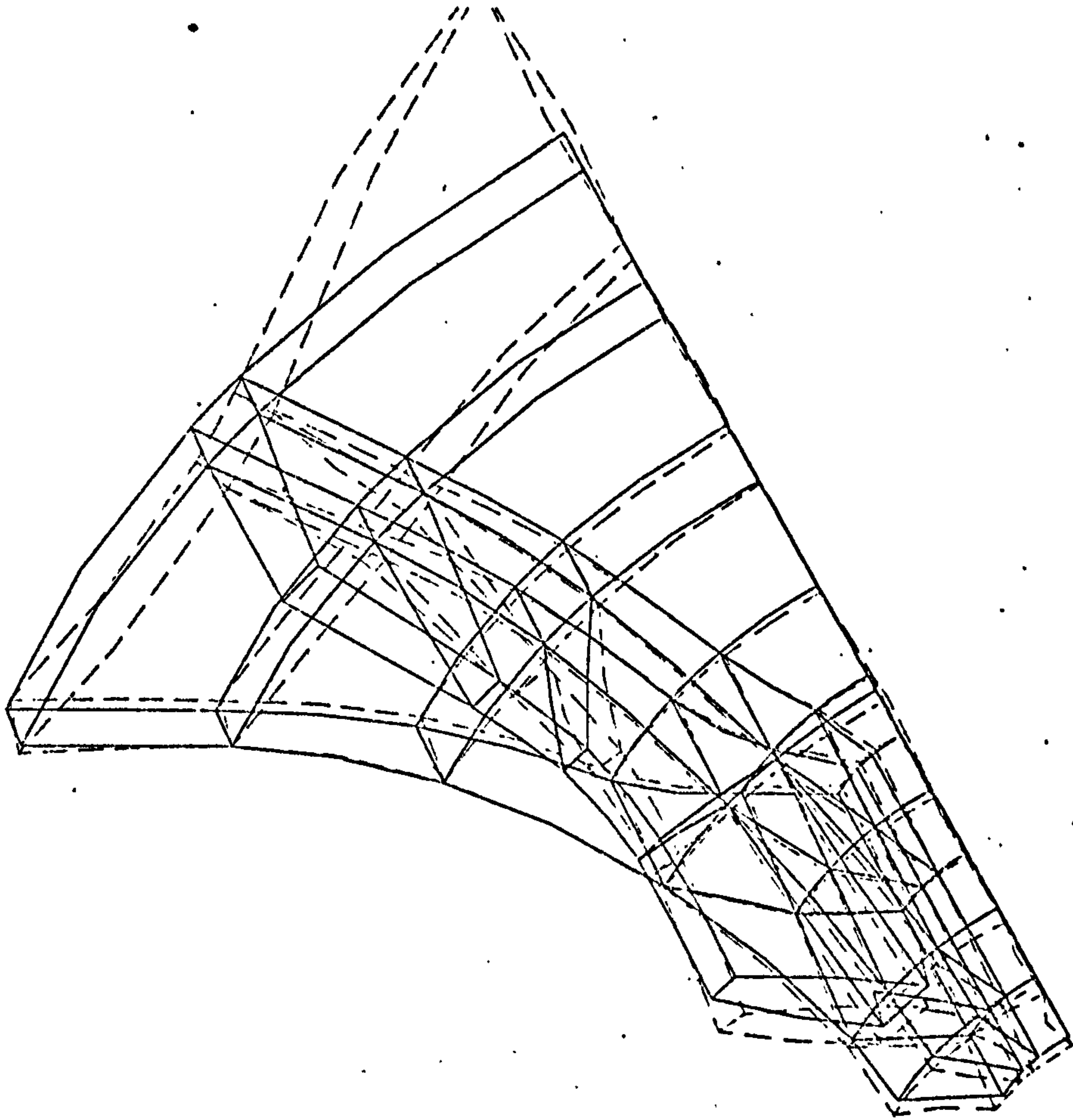


EIGEN MODE FOR HARMONIC NUMBER-2 FREQUENCY IN CYCLES/SEC = 3127.5

3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN IN SCALE: 55.00 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-P MESH-1 8 BLADES

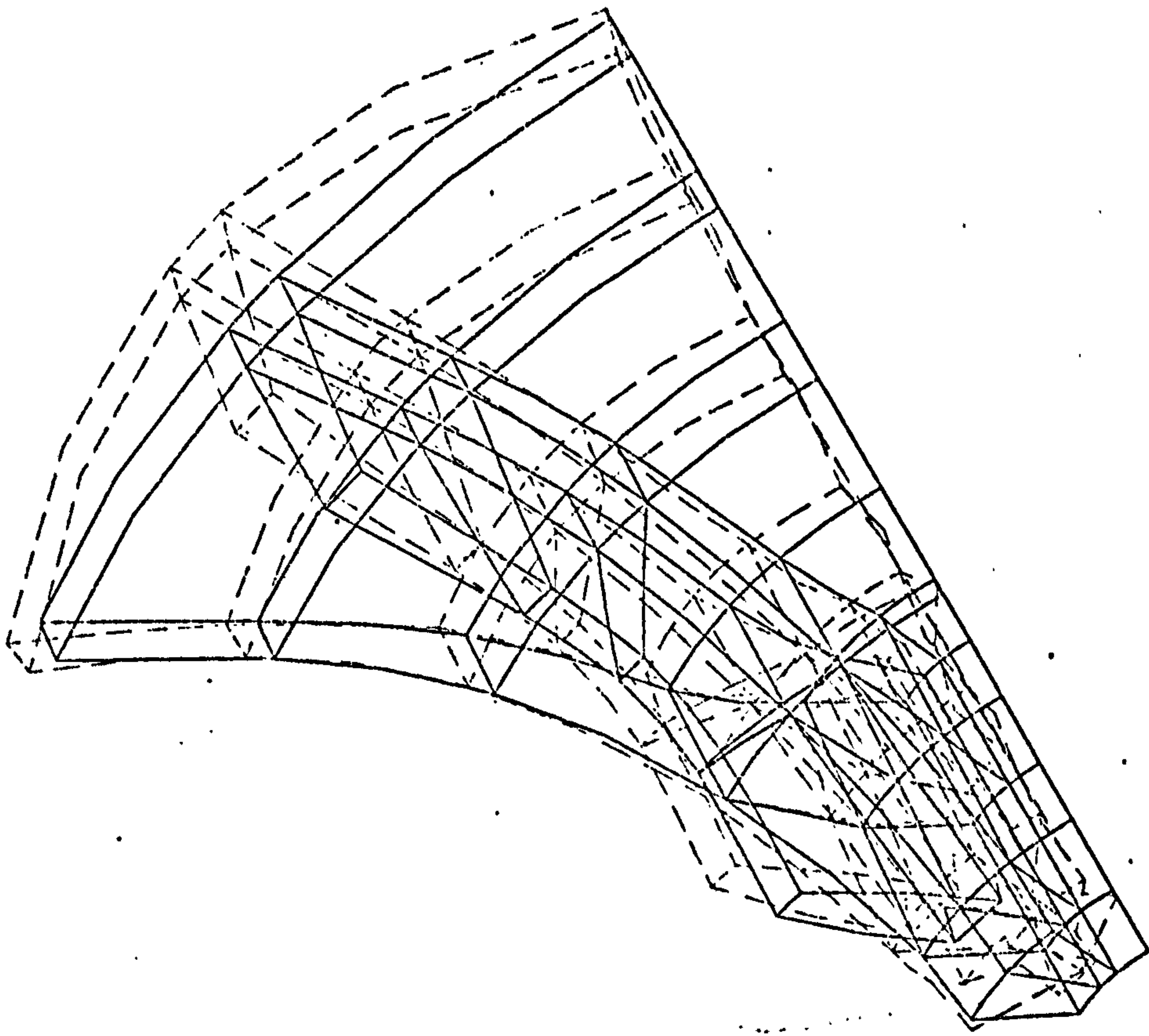
FIG 5-3(g)



EIGEN MODE FOR HARMONIC NUMBER-2 FREQUENCY IN CYCLES/SEC = 8575.3

3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN IN SCALE: 55.00 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-P MESH-1 8 BLADES

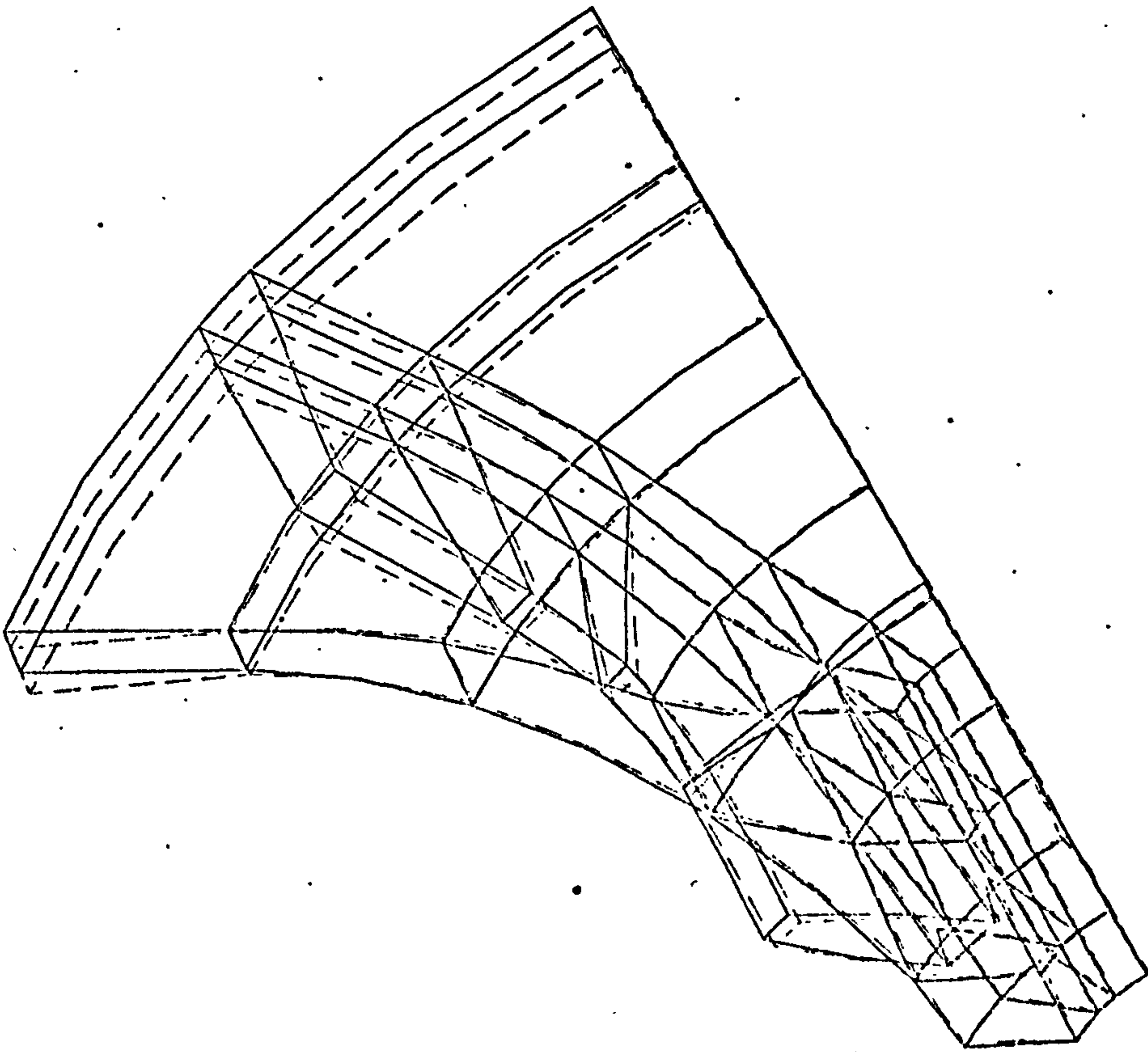


EIGEN MODE FOR HARMONIC NUMBER-2 FREQUENCY IN CYCLES/SEC = 11638.8

3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
CRAN IN IN SCALE: 55.82 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-P MESH-1 8 BLADES

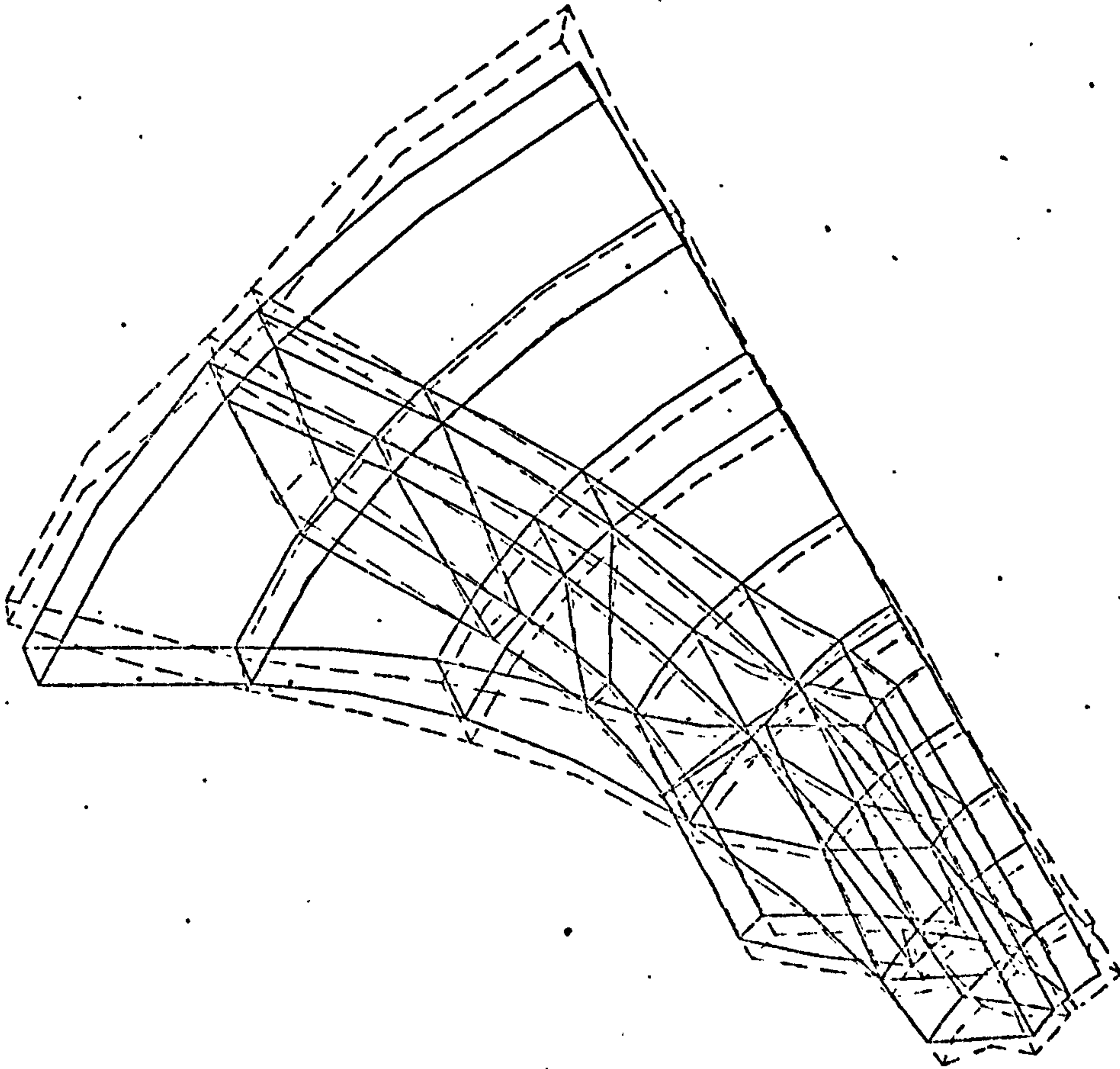
FIG 5-3 (h)



EIGEN MODE FOR HARMONIC NUMBER-2 FREQUENCY IN CYCLES/SEC = 9342.5

3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
CRAN IN IN SCALE: 55.82 TIMES THE SUPPLIED UNITS

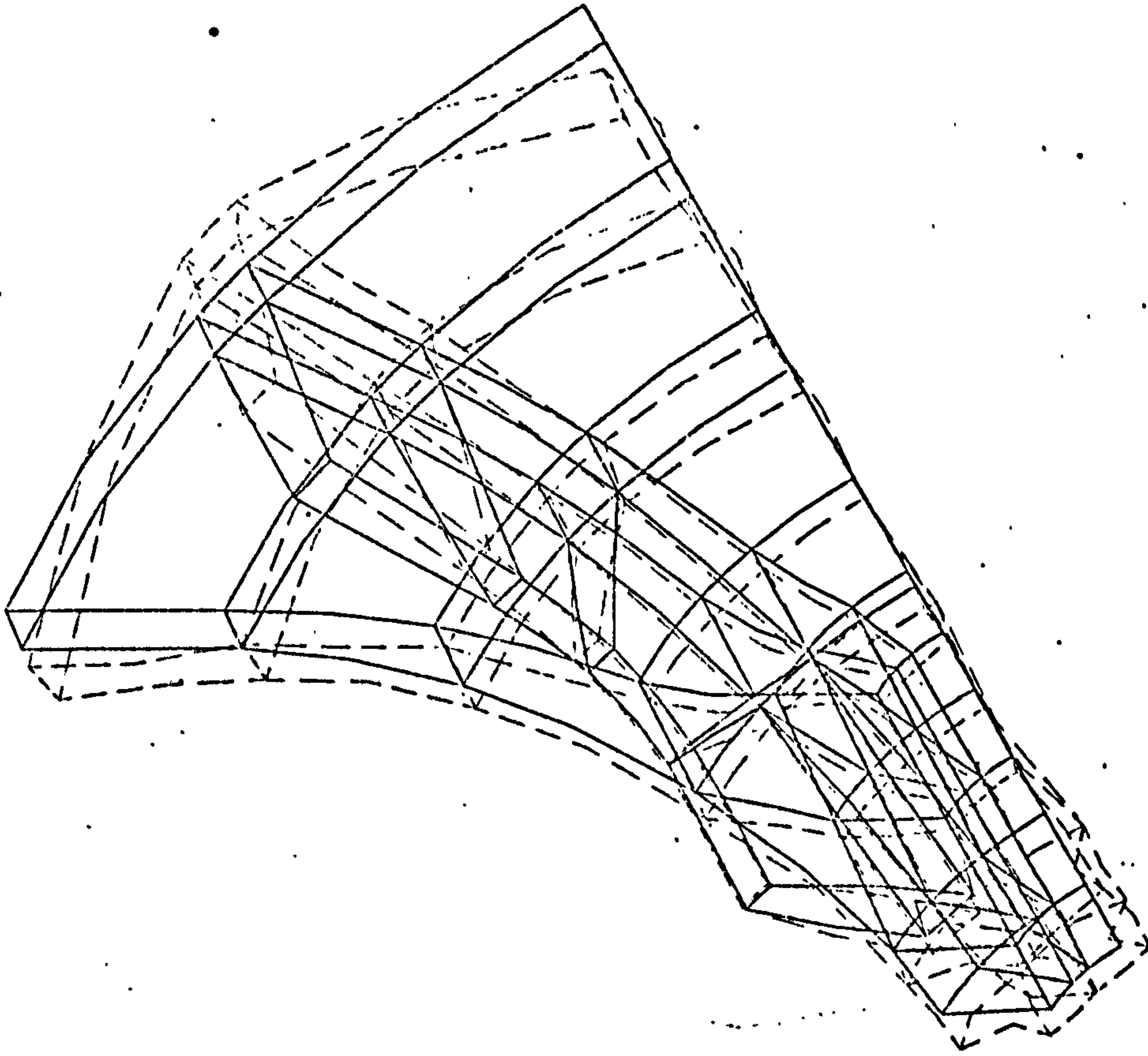
IMPELLER MODEL-P MESH-1 8 BLADES



EIGEN MODE FOR HARMONIC NUMBER-2 FREQUENCY IN CYCLES/SEC = 15674.9

3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN IN SCALE: 55.00 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-P MESH-1 8 BLADES

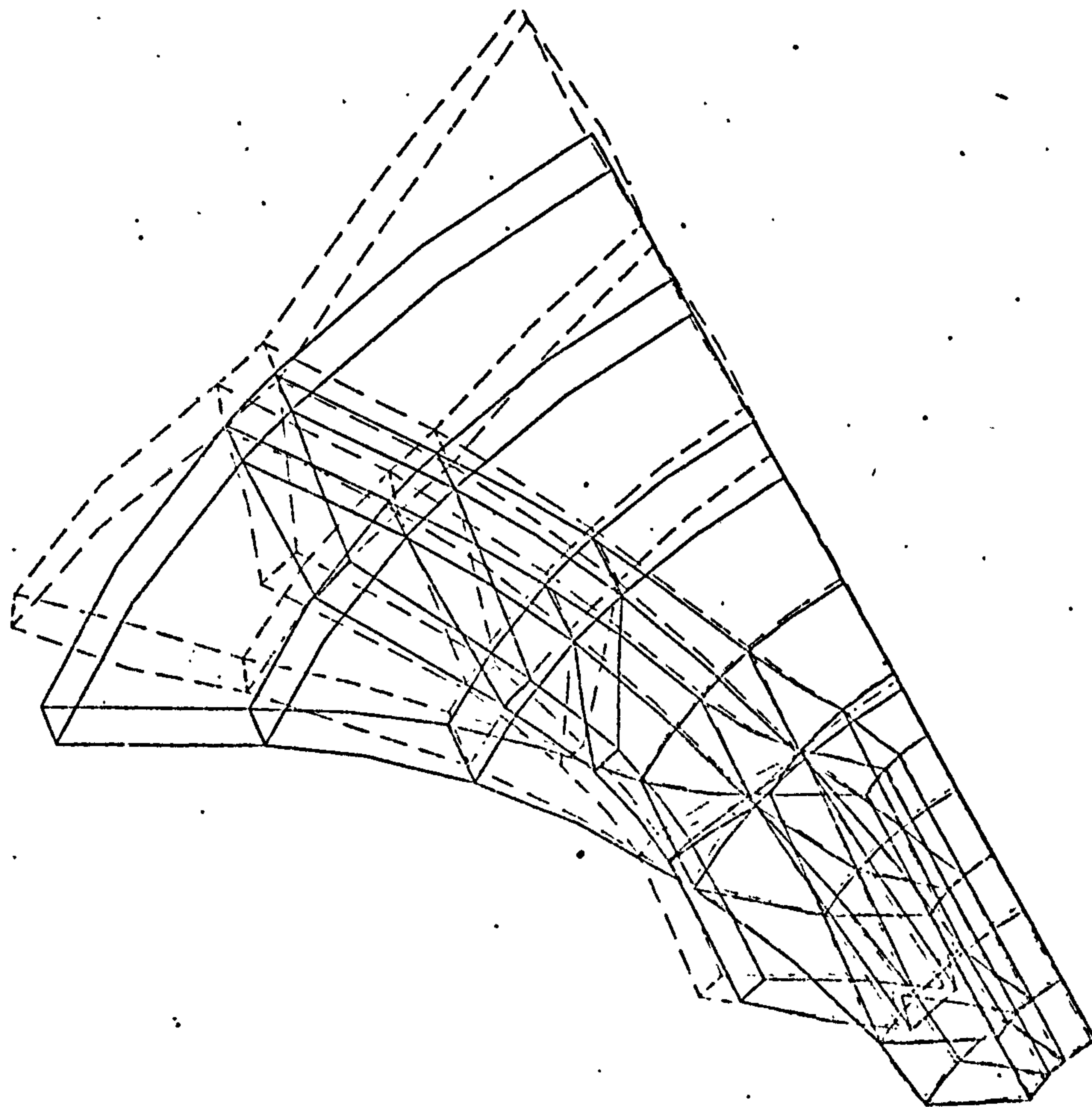


EIGEN MODE FOR HARMONIC NUMBER-2 FREQUENCY IN CYCLES/SEC = 17623.6

3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN IN SCALE: 55.00 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-P MESH-1 8 BLADES

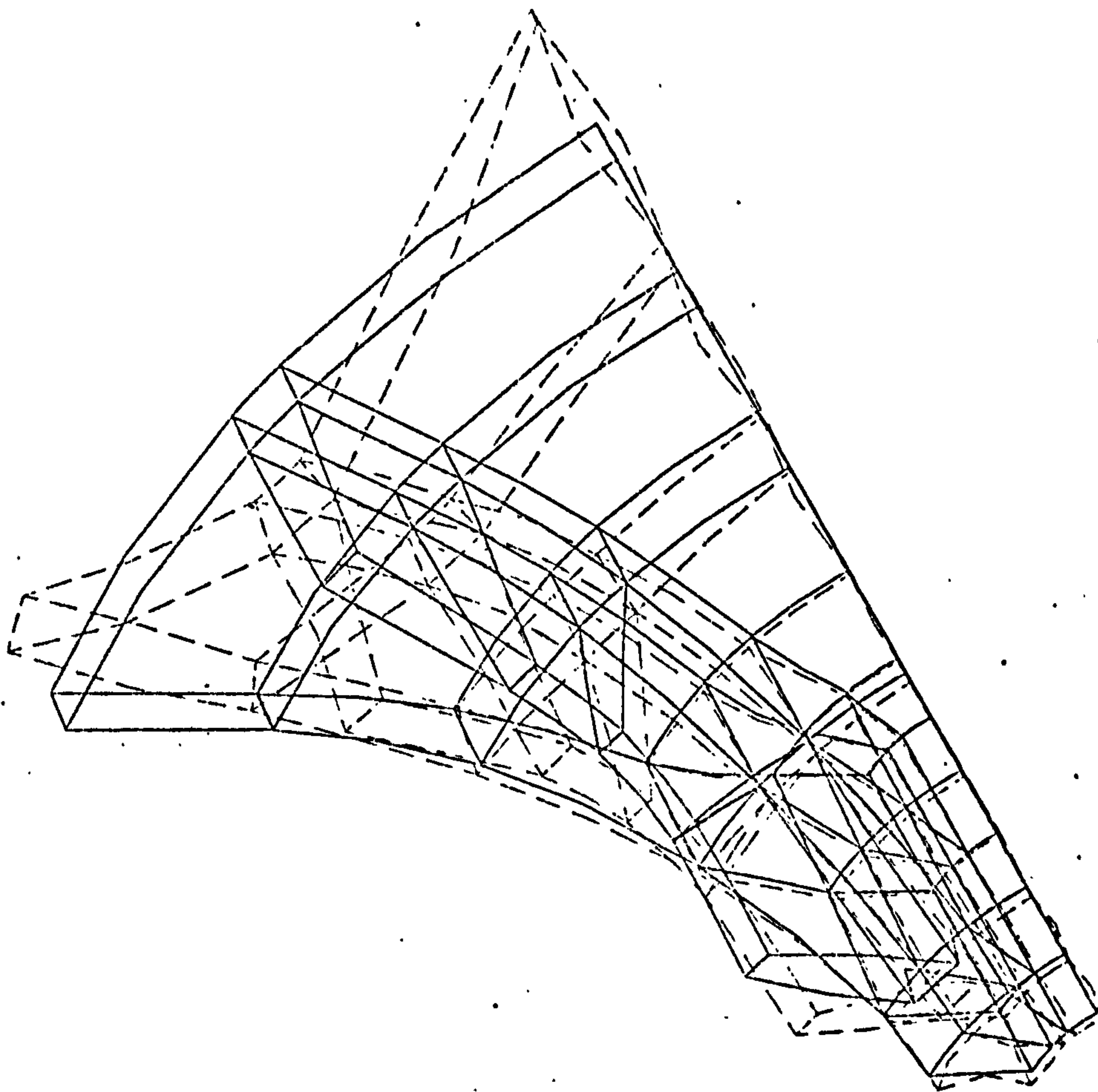
FIG 5-3(ii)



EIGEN MODE FOR HARMONIC NUMBER-3 FREQUENCY IN CYCLES/SEC = 4966.6

3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN IN SCALE: 55.00 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-P MESH-I 8 BLADES

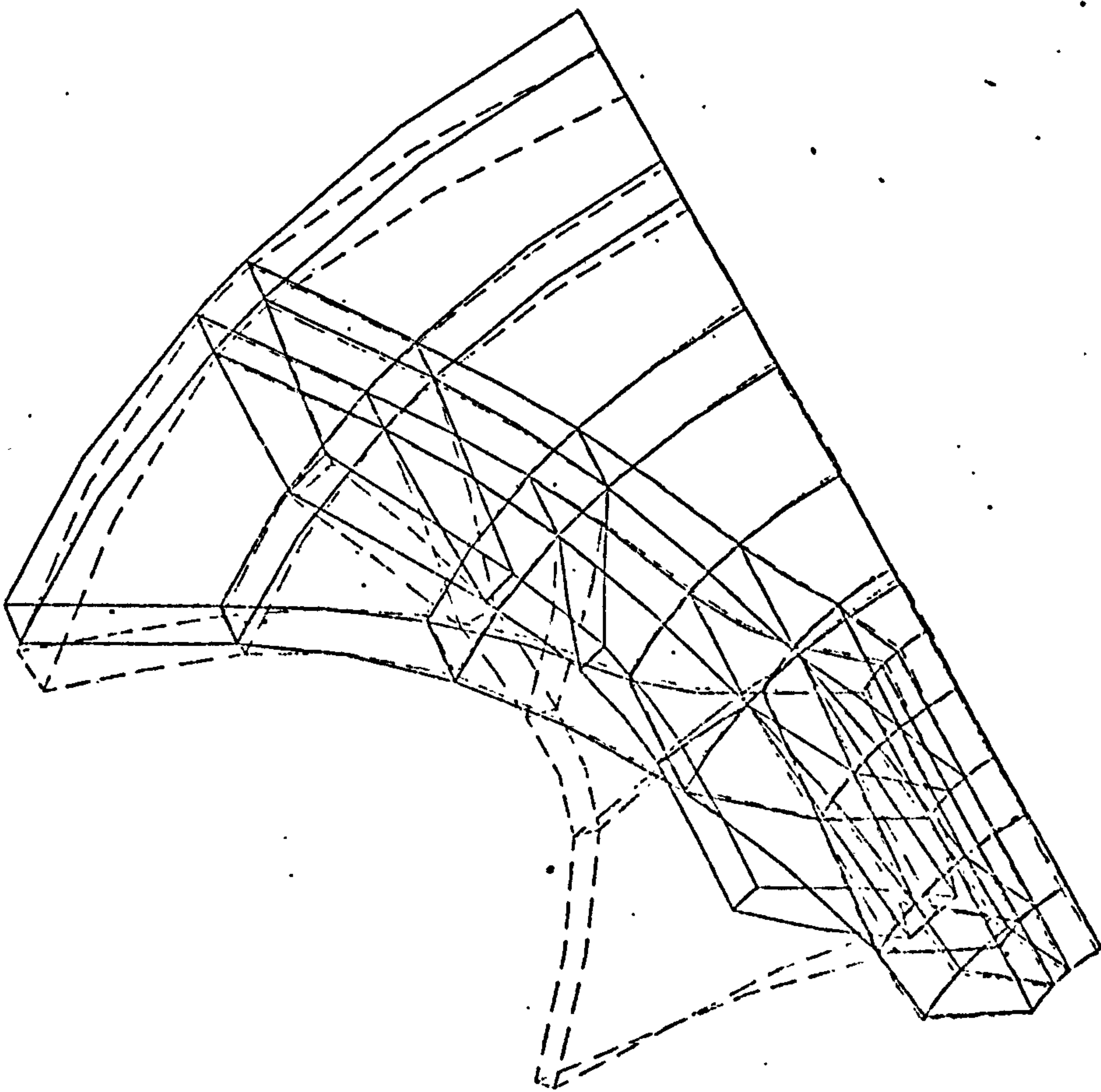


EIGEN MODE FOR HARMONIC NUMBER-3 FREQUENCY IN CYCLES/SEC = 8445.5

3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN IN SCALE: 55.00 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-P MESH-I 8 BLADES

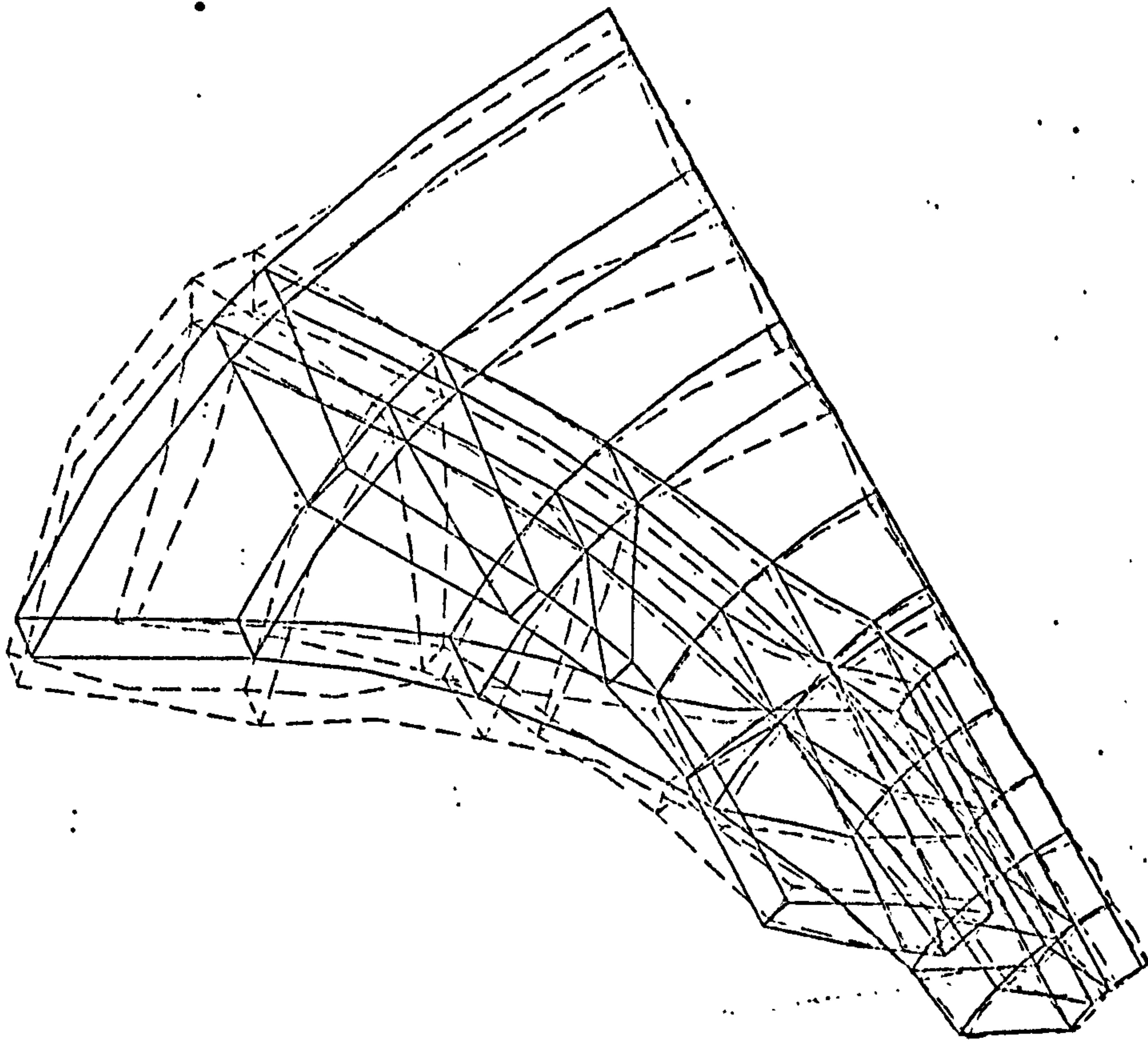
FIG 5-3 (j)



EIGEN MODE FOR HARMONIC NUMBER-3 FREQUENCY IN CYCLES/SEC = 5593.2

3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN IN SCALE: 55.00 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-P MESH-I 8 BLADES

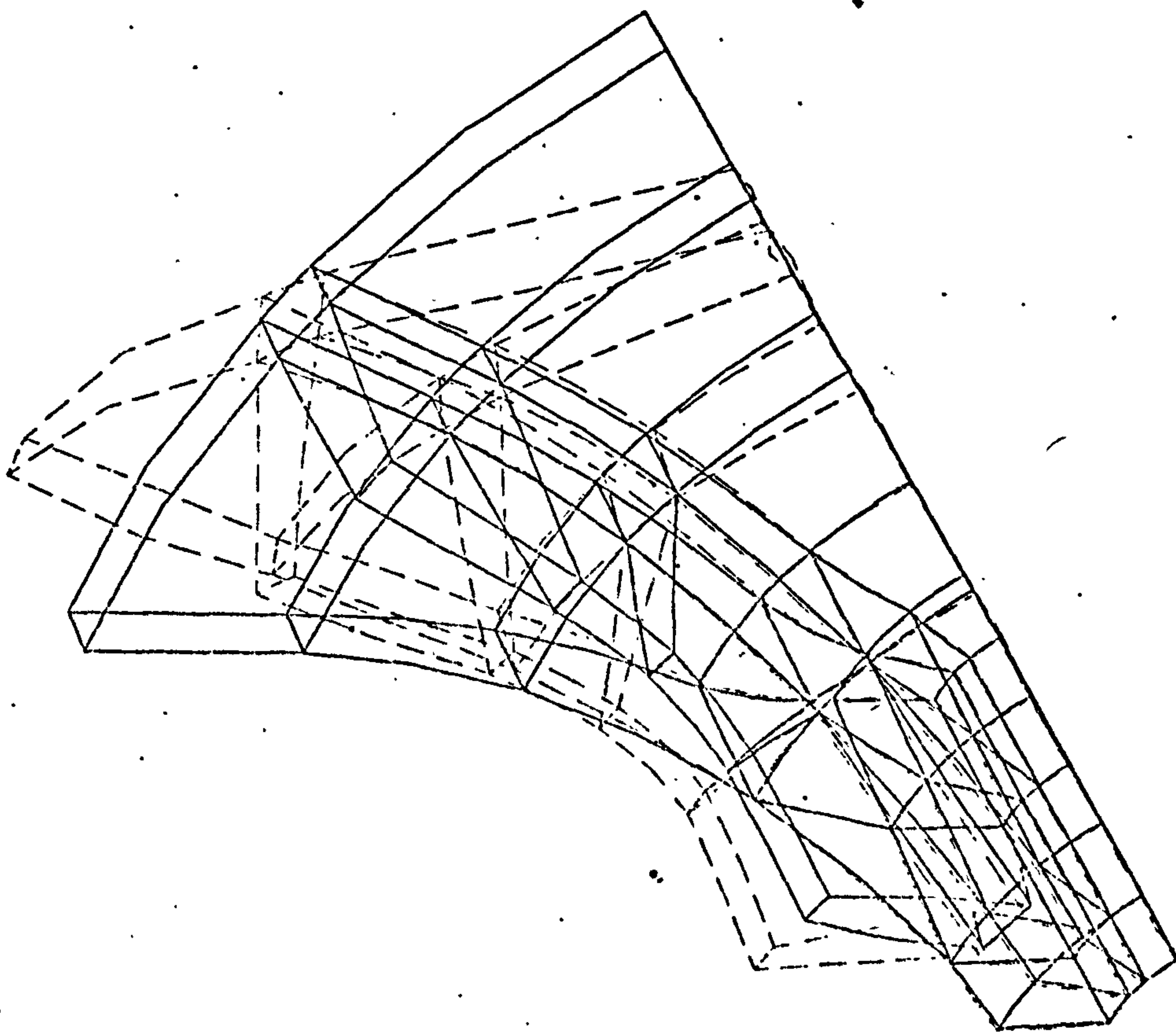


EIGEN MODE FOR HARMONIC NUMBER-3 FREQUENCY IN CYCLES/SEC = 15599.2

3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN IN SCALE: 55.00 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-P MESH-I 8 BLADES

FIG 5-3 (k)

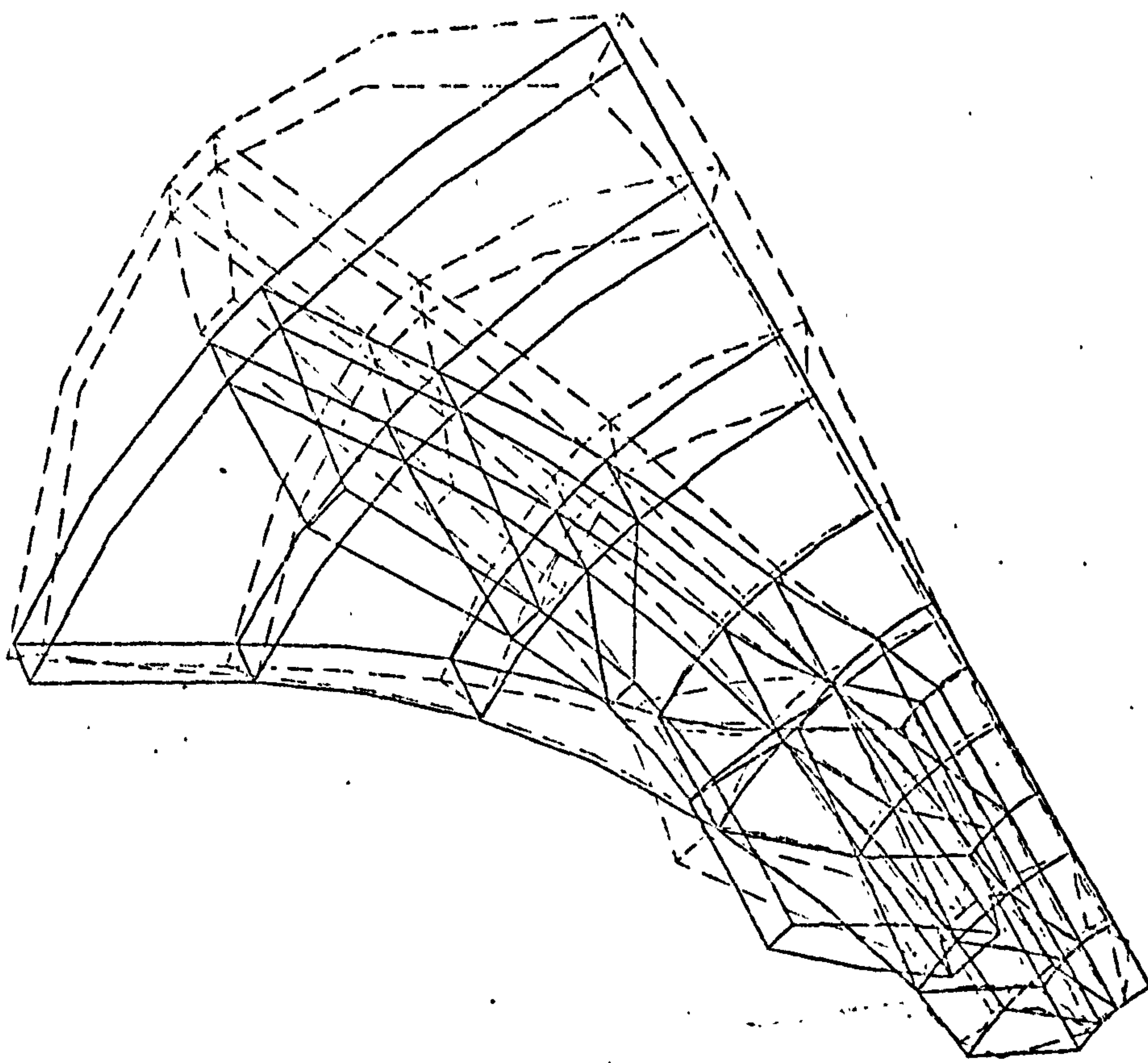


EIGEN MODE FOR HARMONIC NUMBER-4 FREQUENCY IN CYCLES/SEC = 5321.0

3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN IN SCALE: 55.03 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-P MESH-I 8 BLADES

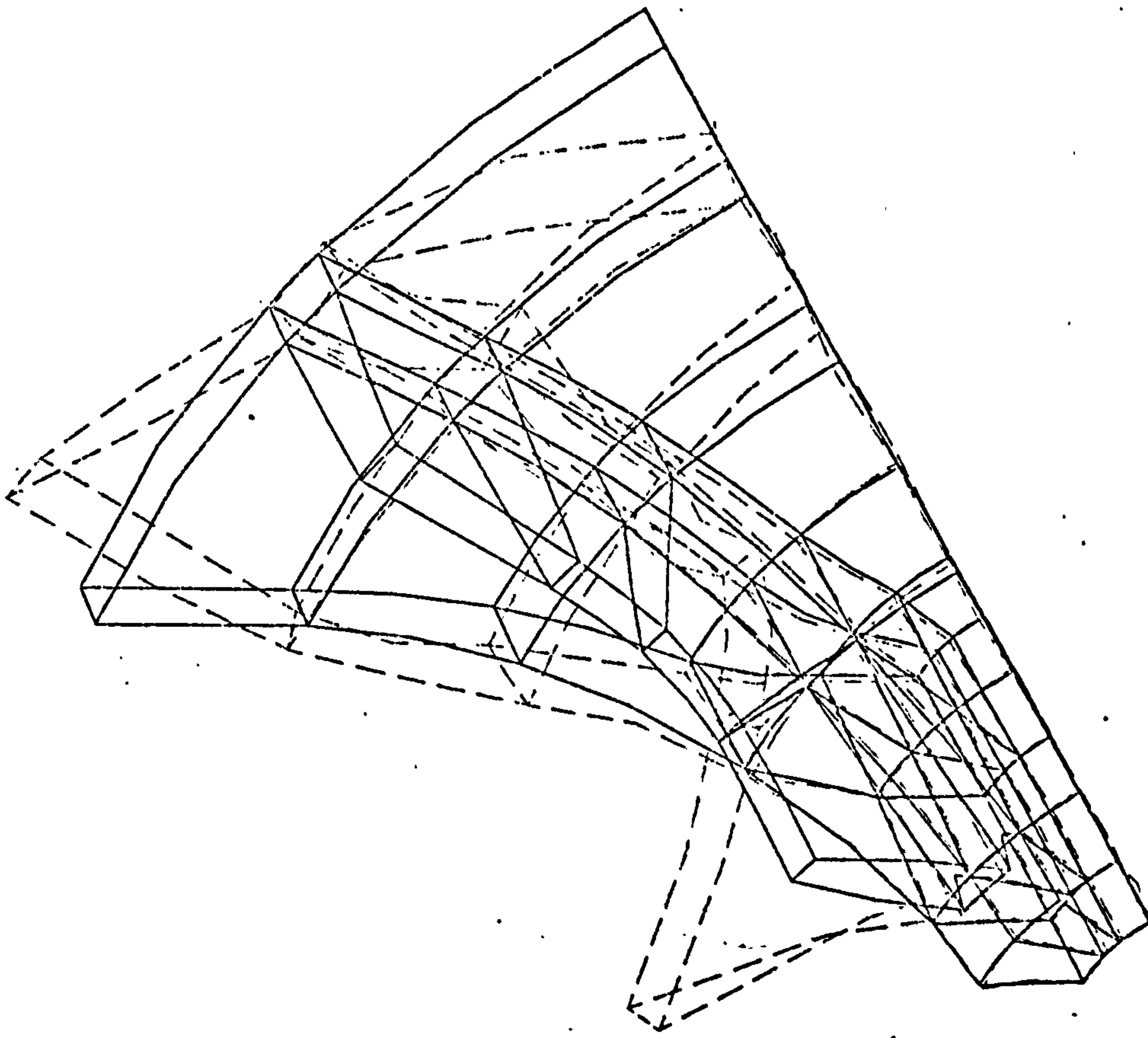
FIG 5-3 (I)



EIGEN MODE FOR HARMONIC NUMBER-4 FREQUENCY IN CYCLES/SEC = 5335.2

3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN IN SCALE: 55.02 TIMES THE SUPPLIED UNITS

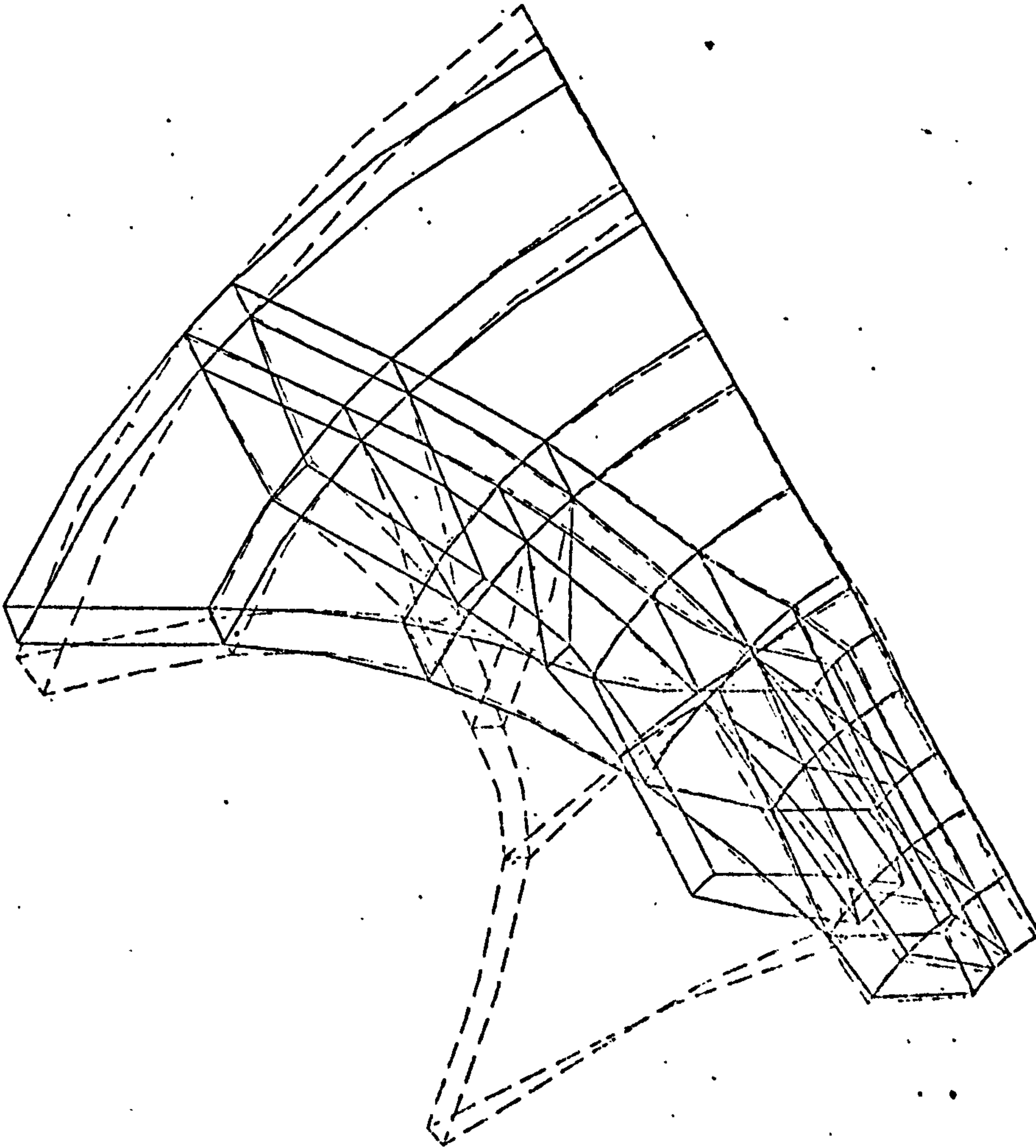
IMPELLER MODEL-P MESH-I 8 BLADES



EIGEN MODE FOR HARMONIC NUMBER-4 FREQUENCY IN CYCLES/SEC = 17478.3

3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN IN SCALE: 55.00 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-P MESH-I 8 BLADES



EIGEN MODE FOR HARMONIC NUMBER-4 FREQUENCY IN CYCLES/SEC = 9646.9

3-D FINITE ELEMENT MODEL OF IMPELLER BLADE SEGMENT
DRAWN IN IN SCALE: 55.00 TIMES THE SUPPLIED UNITS

IMPELLER MODEL-P MESH-I 8 BLADES

FIG 5-3 (m)

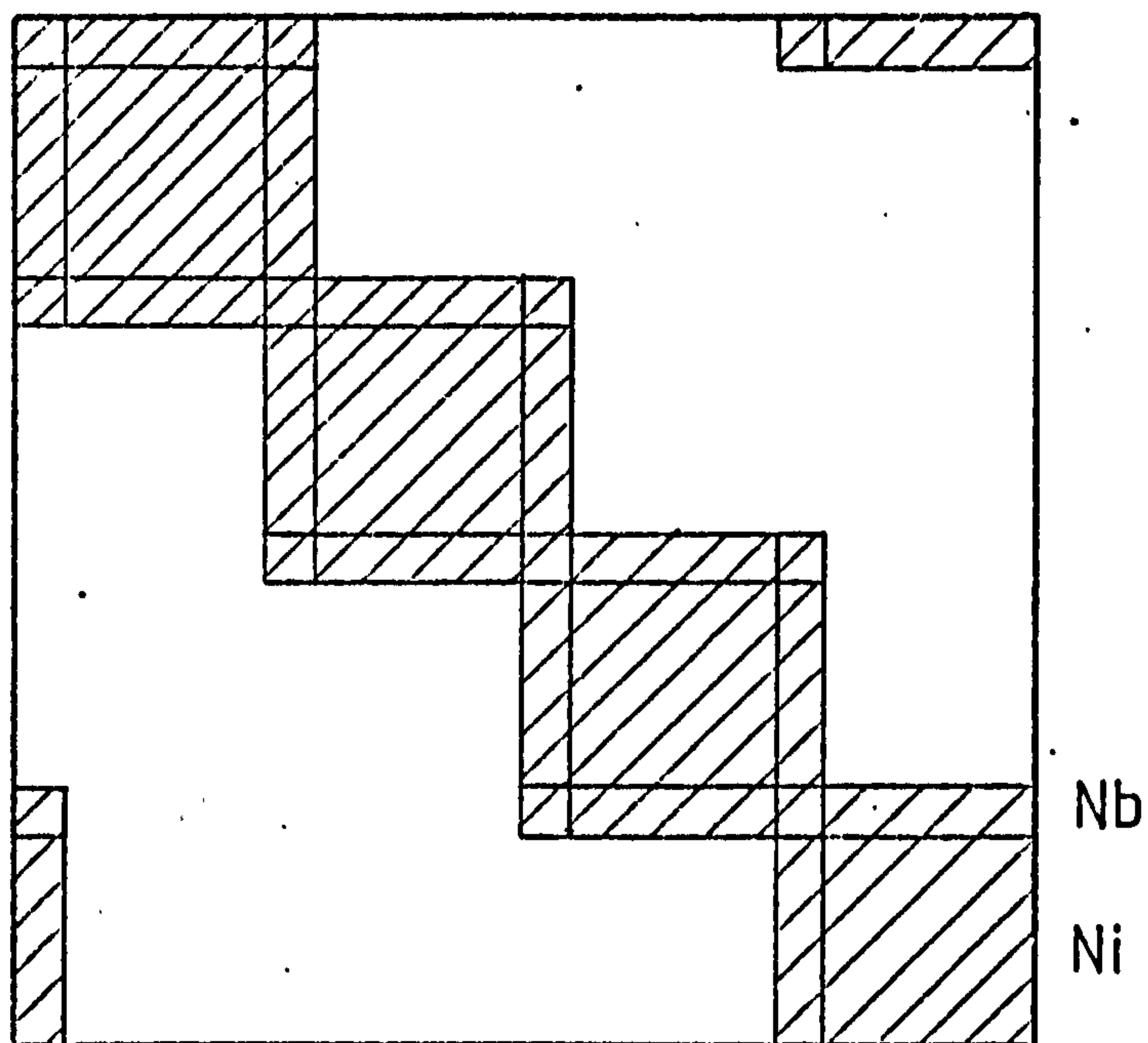


FIG A.1 General Form of Stiffness and Mass Matrices of Typical Structure with Four Repeated Sub-Structures.

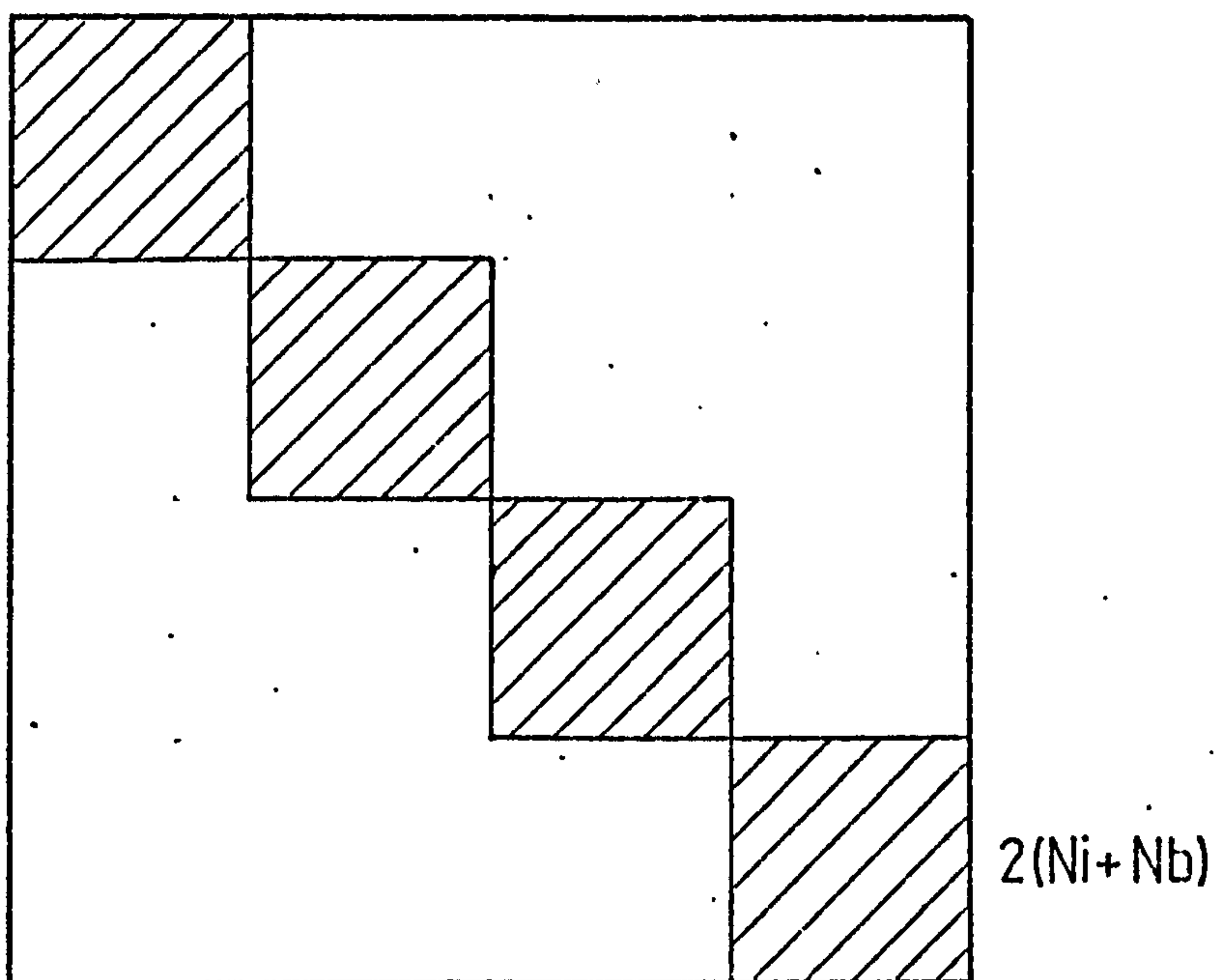


FIG A.2 Form of the Decoupled Stiffness or Mass Matrix after a Fourier Transformation.

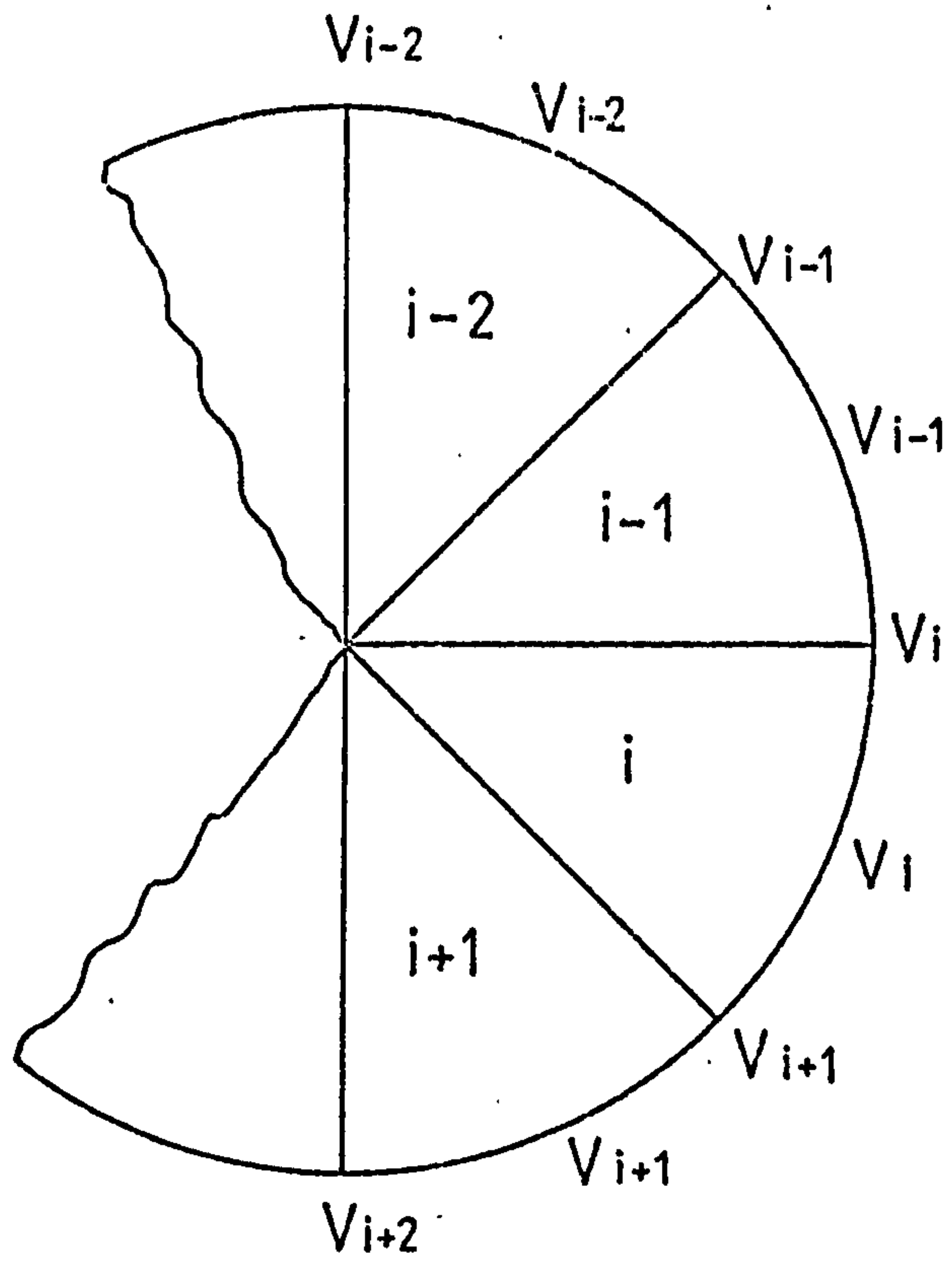


FIG A.3 Cyclically Repetitive Structure.

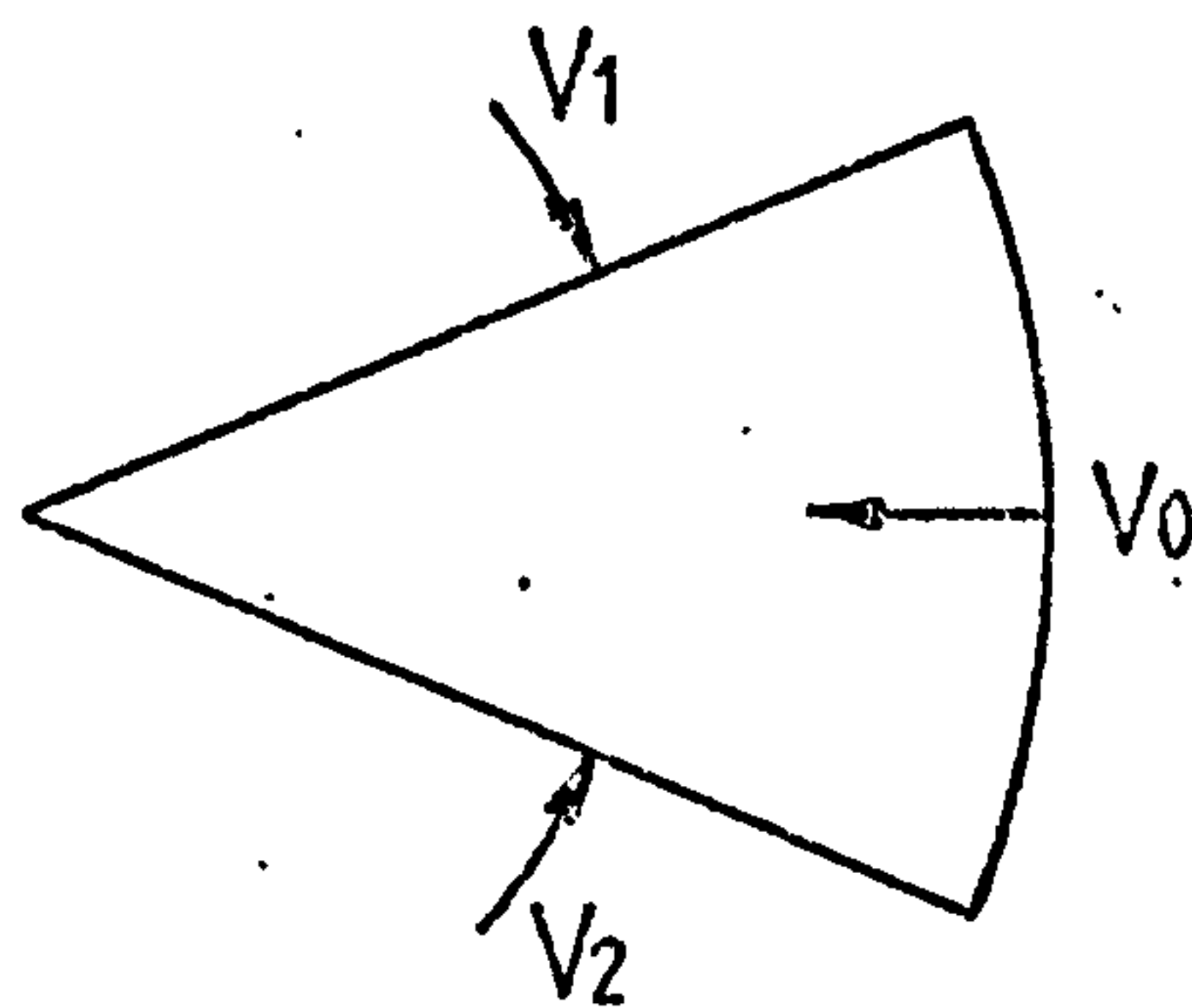


FIG A.4 A Typical Repetitive Sub-Structure.

