Image Cover Sheet

CLASSIFICATION	SYSTEM NUMBER 516688
UNCLASSIFIED	

TITLE

PVAST propeller vibration and strength analysis program version 7.3 user's manual

System Number:

Patron Number:

Requester:

Notes:

DSIS Use only:

Deliver to: CL

This page is left blank

This page is left blank

Copy No. 10

DEFENCE R&D DÉFENSE

PVAST PROPELLER VIBRATION AND STRENGTH ANALYSIS PROGRAM VERSION 7.3 USER'S MANUAL

Koko, T.S., Palmeter, M.F., Chernuka, M.W.
MARTEC Limited
1888 Brunswick Street, Suite 400
Halifax, Nova Scotia, Canada, B3J 3J8

Contract No. W7707-7-4689/001/HAL

Defence R&D Canada

DEFENCE RESEARCH ESTABLISHMENT ATLANTIC

Contractor Report
DREA CR 2000-152
March 2001

National Defence · Défense nationale

Canada

REPRODUCTION QUALITY NOTICE

This document is the best quality available. The copy furnished to DRDCIM contained pages that may have the following quality problems:

: Pages smaller or Larger than normal

: Pages with background colour or light coloured printing

: Pages with small type or poor printing; and or

: Pages with continuous tone material or colour photographs

Due to various output media available these conditions may or may not cause poor legibility in the hardcopy output you receive.

☒ If this block is checked, the copy furnished to DRDCIM contained pages with colour printing, that when reproduced in Black and White, may change detail of the original copy.

Copy No: ________

PVAST Propeller Vibration and Strength Analysis Program Version 7.3 User's Manual

Koko,T S , Palmeter, M F , Chernuka, M.W.
MARTEC Limited
1888 Brunswick Street, Suite 400
Halifax, Nova Scotia, Canada, B3J 3J8

Contract No W7707-7-4689/001/HAL

Defence Research Establishment Atlantic

Contractor Report
DREA CR 2000-152
March 2001

Authors

Koko,T.S., Palmeter, M.F., Chernuka, M.W.

Contract Scientific Authority

D.C.Stredulinsky

Approved by

J.L. Kennedy

Head, Warship Signatures & Safety

Approved for release by

Kirk Foster

Chair/DRP

The scientific or technical validity of this Contract Report is entirely the responsibility of the contractor and the contents do not necessarily have the approval or endorsement of Defence R&D Canada.

Abstract

This report is a user's manual for the finite element analysis code called PVAST (Propeller Vibration And STrength) that is used for prediction of stress and vibration in marine propellers. PVAST can automatically generate propeller finite element models from basic propeller geometry defined by the orientation of 2D "wrapped" blade sections at specified radii. A number of different models can be considered including; a single blade, a single blade with fillet, a blade-fillet-palm model, a blade-fillet-hub segment model, and multiple blade and hub model. The types of structural finite element analyses that can be conducted include; static analysis with user-defined blade pressure distributions and point loads, natural frequency analysis in air and in water, time domain analysis with applied loads and support motion, response spectrum analysis and frequency response analysis. The program provides 2D and 3D plotting of blade geometry, finite element models and post-processing to provide visualization of predicted finite element model displacements, stresses and mode shapes. The code has been modified recently to include a "Windows" graphical user interface implemented with a combination of GKS and MS Visual C++. This provides a more "friendly" user interface and is a major change in the look and feel of PVAST Version 7.3 compared with earlier versions of the code.

Résumé

Le rapport est un manuel d'utilisation pour le code d'analyse par éléments finis appelé PVAST (Propeller Vibration And Strength [vibrations et résistance des hélices]) qui sert à prédire les contraintes et les vibrations sur les hélices de navires. Ce logiciel peut générer automatiquement des modèles par éléments finis d'hélices à partir de leur géométrie de base définie par l'orientation des sections de pales en 2 dimensions à des rayons spécifiés. Un certain nombre de modèles peuvent être étudiés : une seule pale, une seule pale avec raccordement de racine, un modèle de pale-raccordement-tourteau, un modèle de pale-raccordement-moyeu et un modèle avec pales multiples et un moyeu. Les types d'analyse structuralle par éléments finis qui peuvent être exécutées comprennent les analyses statiques avec les répartitions des pressions sur les pales et les contraintes ponctuelles définies par l'utilisateur, les analyses des fréquences propres dans l'air et dans l'eau, l'analyse dans le domaine temporel avec les contraintes exercées et les déplacements des supports, l'analyse du spectre de réponses et l'analyse des réponses en fréquences. Le programme donne un tracé de la géométrie des pales en 2 D et en 3 D, des modèles par éléments finis et un post-traitement pour obtenir une visualisation des prévisions par éléments finis des mouvements des modèles, des contraintes et des formes de modes. Le code a été modifié récemment pour inclure une interface graphique de l'utilisateur sur système d'exploitation Windows; elle fonctionne avec une combinaison d'un système graphique de base GKS et du logiciel MS Visual C++, ce qui donne une interface plus pratique et constitue un changement majeur dans l'apparence et la présentation de la version 7.3 de PVAST par rapport aux versions précédentes de ce code.

This page intentionally left blank.

Executive summary

Background: A marine propeller blade has very a complicated geometry. In order to obtain an accurate assessment of propeller strength and vibration characteristics it is necessary to use numerical analysis methods. Methods for structural analysis of marine propellers have been developed at DREA through in-house and contracted research based on the finite element method. A computer code called PVAST (Propeller Vibration And STrength) has been written to automatically generate propeller finite element models from basic propeller geometric data. The code can automatically produce a number of different models including; a single blade, a single blade with fillet, a blade-fillet-palm model, a blade-fillet-hub segment model, and a multiple blade and hub model. The types of structural finite element analyses that can be conducted include; static analyses with user-defined blade pressure distributions and point loads, natural frequency analyses in air and in water, time domain analyses with applied loads and support motion, response spectrum analyses and frequency response analyses. The program provides 2D and 3D plotting of blade geometry, finite element models and post-processing visualization of predicted finite element model displacements, stresses and mode shapes. The code has been modified recently to include a "Windows" graphical user interface implemented with a combination of GKS and MS Visual C++ coding.

Principal Results: The principal result arising from the recent work on the code is a user-friendly graphical interface. This is a major change in the look and feel of PVAST Version 7.3 compared with earlier versions of the code. It is easier to use and accessible to more users since it can now be run on PCs with Windows95, 98 and NT operating systems, whereas previously, it could only be run on a mainframe computer or UNIX work station.

Significance of Results: Because of their complicated three-dimensional geometry, the generation of propeller finite element models using general purpose finite element modeling programs is very time consuming and prone to errors. The PVAST code allows the user to enter propeller geometry in formats commonly used by propeller designers. It provides a number of graphical tools for checking the input geometry and automatically generating finite element models. It also presents results in many formats, many specialized for propeller geometry. This program will allow DREA scientists and navy engineers to quickly build reliable structural models of propellers to analyze strength and vibration problems in existing designs and also to assess structural characteristics of future designs.

Koko,T.S., Palmeter, M.F., Chernuka, M.W. 2001. PVAST Propeller Vibration and Strength Analysis Program Version 7.3 User's Manual. DREA CR 2000-152 Defence Research Establishment Atlantic.

Sommaire

Explication: Une hélice de navire possède une géométrie très complexe. Pour obtenir une évaluation précise des caractéristiques de résistance et de vibration d'une hélice, il est nécessaire de faire appel aux méthodes d'analyse numériques. Les méthodes d'analyse structurale des hélices de navires ont été développées au CRDA grâce des recherches effectuées sur place ou sous contrat, toutes fondées sur la méthode par éléments finis. Un code informatique appelé PVAST (Propeller Vibration And STrength [vibrations et résistance de hélices]) a été développé pour générer automatiquement des modèles par éléments finis d'hélices à partir de données géométriques d'hélices de base. Ce code peut produire automatiquement un certain nombre de modèles différents : une seule pale, une seule pale avec raccordement de racine, un modèle de pale-raccordement-tourteau, un modèle de pale-raccordement-moyeu et un modèle avec pales multiples et moyeu. Les types d'analyse structurale par éléments finis qui peuvent être exécutées comprennent : les analyses statiques avec répartitions des pressions sur les pales et les contraintes ponctuelles définies par l'utilisateur, des analyses des fréquences propres dans l'air et dans l'eau, des analyses dans le domaine temporel appliquées aux contraintes et aux mouvements des supports, des analyses du spectre des réactions et des analyses des réponses en fréquences. Le programme donne un tracé de la géométrie des pales en 2 D et en 3 D par éléments finis, des modèles par éléments finis et une visualisation post-traitement des prédictions des mouvements des modèles, des contraintes et des formes de mode. Le code a été modifié récemment pour comprendre une interface graphique avec l'utilisateur sur Windows, mis en œuvre avec une combinaison d'un système graphique de base GKS et du logiciel MS Visual C++.

Principaux résultats: Le principal résultat des travaux récents effectués sur le code est une interface graphique plus pratique. Il s'agit d'un changement important dans l'apparence et la présentation de la version 7.3 de PVAST, par rapport aux versions précédentes de ce code. Il est plus facile à utiliser et est accessible à plus d'utilisateurS car il peut maintenant fonctionner sur des ordinateurs personnels avec les systèmes d'exploitation Windows 95, 98 and NT, alors qu'auparavant il ne pouvait fonctionner que sur un ordinateur central ou un poste de travail UNIX.

Importance des résultats: En raison de leur géométrie tri-dimensionnelle complexe, la génération de modèles d'hélice par des programmes universels de modélisation par éléments finis demande beaucoup de temps et est sujette à des erreurs. Le code PVAST permet à l'utilisateur d'entrer une géométrie d'hélice dans des formats fréquemment exploités par des concepteurs d'hélices. Il fournit un certain nombre d'outils graphiques pour vérifier la géométrie retenue et pour générer automatiquement des modèles par éléments finis. Il présente aussi les résultats en plusieurs formats, beaucoup d'entre eux spécialisés dans la géométrie des hélices. Ce programme permettra aux scientifiques du CRDA et aux ingénieurs maritimes de construire rapidement des modèles de structure d'hélices fiables pour analyser les problèmes de vibration et de résistance des hélices existantes et aussi d'évaluer les caractéristiques structurales des hélices futures.

Koko,T.S., Palmeter, M.F., Chernuka, M.W. 2000. Manuel D'utilisation Du Programme D'analyse Des Vibrations Et De La Résistance Des Hélices PVAST, Version 7.3. CRDA CR 2000-152 Centre pour la Recherche de la D'efence Atlantique.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

1. INTRODUCTION

1.1 Background

PVAST is a computer program for the strength and vibration analysis of marine propellers. It is based on the finite element method of structural analysis and is designed to automatically generate finite element models for a wide variety of propeller shapes. The three-dimensional spatial co-ordinates of the blade geometry are automatically generated from the two-dimensional blade section data format in which the blade geometry is generally described. Finite element meshes of the blade geometry are automatically generated from a choice of shell and solid elements. The program can graphically display the geometry and the finite element model for data checking purposes. Static, natural frequency and dynamic response analyses can be carried out with the program. The analytical results can be processed and displayed in graphic form for ease of interpretation to show stresses, displacements, natural frequency of vibration and dynamic response.

1.2 PVAST History

The PVAST program is developed, maintained and marketed by Martec Limited. The first version of PVAST was produced in 1975 for operation on the DEC and VAX/VMS operating systems [1]. At that time the three-dimensional spatial geometry, and hydrodynamic loads were generated by separate programs called BLADE [2] and PUF2 [3] that were developed by the Defence Research Establishment Atlantic (DREA). These two programs generated blade section geometry and load data files (with extension names "T16" and "LIN") that were required by PVAST for generating the blade finite element model. Several modifications have been made over the years to improve the modelling capabilities and to keep pace with technological developments, resulting in several versions of the program.

PVAST Versions 1 to 4 were designed to operate on the DEC and VAX/VMS systems [4-5]. In those versions, user interaction with the program was mainly via scrolling menus. Also, the program was operated in a modular fashion where individual modules (or programs) were executed one at a time or the modules were organized in a series of overlaid programs with predefined processing steps. The various modelling options were identified by the subprogram names.

Efforts to use modern GUI and graphics plotting capabilities and to port the program to the PC and UNIX environments began with Version 6 of the program [6]. In Version 6, a graphical user interface was implemented to improve the user interaction. The GUI and graphics capabilities were developed with the Graphical Kernel System (GKS) to permit cross-platform compatibility. The need to develop a platform independent code compromised the quality of the GUI and graphics capabilities, and besides, the GUI was not completely implemented in Version 6. This gave rise to the current version.

This current version of PVAST is identified as Version 7.3, [7] to be consistent with the version of the VAST finite element solver with which it is compatible. In this version, the GUI is fully implemented and the program is designed to be menu driven with all the modules integrated in a seamless fashion. The GUI and graphical capabilities are derived from the MGDSA graphics program [8]. The MGDSA GUI and graphics capabilities are derived from a combination of GKS and MS Visual C++. MGDSA provides general pre- and post-processing capabilities for the VAST suite of programs [8]. However, special propeller blade modelling and plotting capabilities that are unique to propeller blades are also provided to complement the general capabilities. There is a major change in the look and feel of PVAST Version 7.3 from earlier versions.

1.3 PVAST Capabilities

PVAST has several capabilities covering the types of structural models, types of analysis and display options. Detailed descriptions of these options are provided in Reference 5 and are summarized below.

1.3.1 Types of Propeller Blade Models

There are eight types of structural models available in PVAST. These include:

- Blade Only
- Blade-Fillet
- Blade-Fillet-Palm (Bolt as Rods)
- Blade-Fillet-Palm (Super Elements)
- Blade-Fillet-Hub
- Multi-Blade Hub
- Key-way
- Air-channel

Figures 1.1 to 1.8 show typical views of the eight structural models.

1.3.2 Analysis Types

PVAST offers the following analysis types:

- Static analysis
- Natural frequency analysis in air
- Natural frequency analysis in water
- Time history dynamic analysis due to applied loading or support motion
- Response spectrum analysis
- Frequency response analysis

1.3.3 Display Options

PVAST provides several plotting options that are categorized into *General Plots* and *Special Plots*. The *General Plots* option utilizes the MGDSA general-purpose pre- and post-processing graphics capabilities for displaying the propeller finite element models and results. The *Special Plots* option utilizes special graphics capabilities for displaying data, models and results that are specific to propeller blades. Table 1.1 shows the display options available in PVAST.

Table 1.1: PVAST Display Options

GENERAL PLOT	SPECIAL PLOTS
Pre-Processing: • Blade, root, substructure and fluid FE modes • FE loads and boundary condition plots	Pre-Processing: • Blade geometry input data • Individual section plots • Magnified nose/tail sections • Views of single blade sections • View of assembled blade • Chordwise pressure plots • Blade-to-root intersection • Fluid FE model
Post-Processing: • Displacements • Stresses • Mode Shapes	Post-Processing • Chordwise displacements and stresses • Blade section displacement and mode shapes • Displacement/stress time histories at selected radial/chordwise positions • Radial, tangential and normal force vectors

1.3.4 Operating System

PVAST is designed to be hardware portable and is available on personal computers (PC) with Windows 95/98/NT operating system.

1.4 Applicable Documents

The following documents should be consulted to gain familiarity with marine propeller blade geometry, finite element theory, and the MGDSA graphics capabilities:

- M.W. Chernuka, M.E. Norwood and T.S. Koko. "Propeller Vibration and Stress Analysis by Finite Element Methods (PVAST) Version 4 User's Manual." Martec Limited, July 1992.

- D.R. Smith and T.R. MacFarlane. "BLADE: A Propeller Blade Geometry Generator Program User's Manual." DREA Technical Communication 91/302, February 1991.

- J.L. Kerwin and C.S. Lee. "Prediction of Steady and Unsteady Marine Propeller Performance by Lifting Surface Theory." Vol. 86, 1978. (Provides background on lifting surface programs PINV4 and PUF2 for steady and unsteady propeller force predictions, respectively. Distribution of these program is limited.)

- MicroVAST Graphics System User's Manual. Martec Limited, March 1996.

- Vibration and Strength Analysis Program (VAST): User's Manual Version 7.3, Martec Limited, April 1997.

1.5 Organization of This Manual

The remainder of this manual is organized as follows:

Chapter 2: Provides an overview of the PVAST program structure in which the menu system is described.
Chapter 3: Details the steps to be taken in using PVAST to develop propeller blade models running analyses, and displaying results.
Chapter 4: Describes, in detail, how to create and modify the propeller blade geometry.
Chapter 5: Describes, in detail, how to create and modify the various propeller blade finite element models.
Chapter 6: Describes how to display the propeller blade geometry and finite element models.
Chapter 7: Provides a description of how to perform a VAST [7] finite element analysis of the propeller blade model.
Chapter 8: Describes how to display the PVAST analysis results.
Chapter 9: Describes tools that are available in MGDSA for manipulating finite element models.

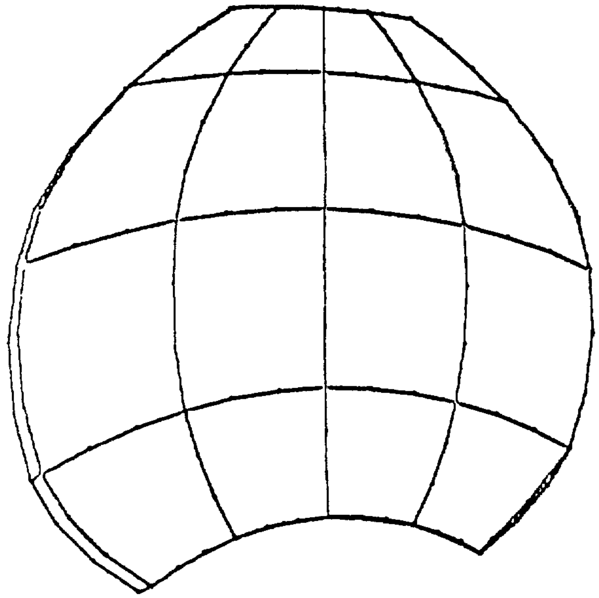


Figure 1.1: Blade Only Model

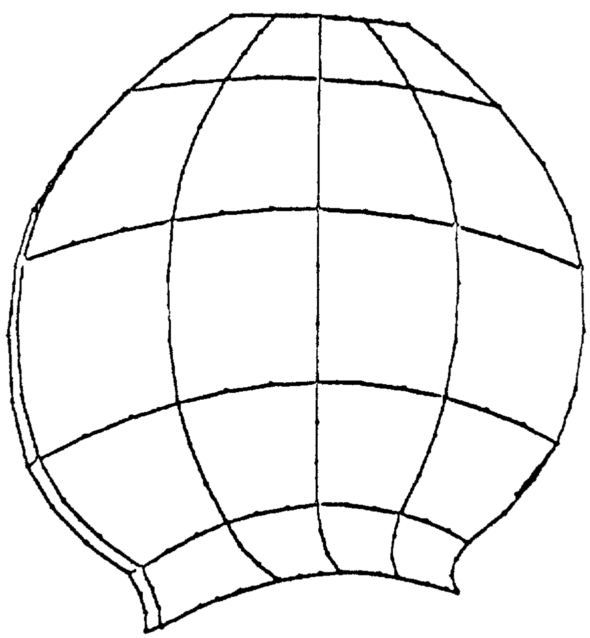


Figure 1.2: Blade-Fillet Model

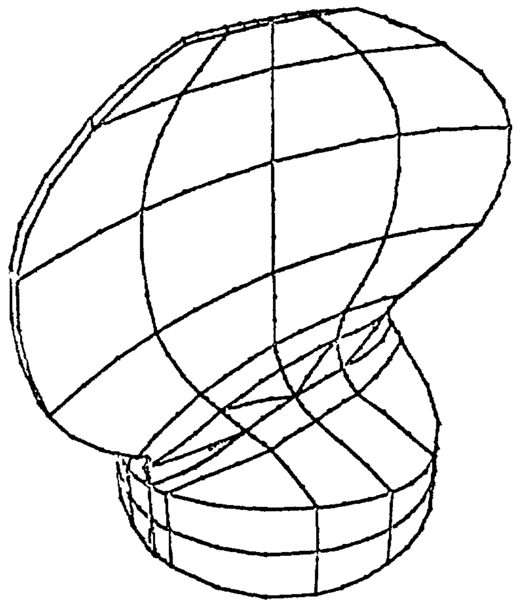


Figure 1.3: Blade-Fillet-Palm (Bolts as Rods) Model (shown without bolts)

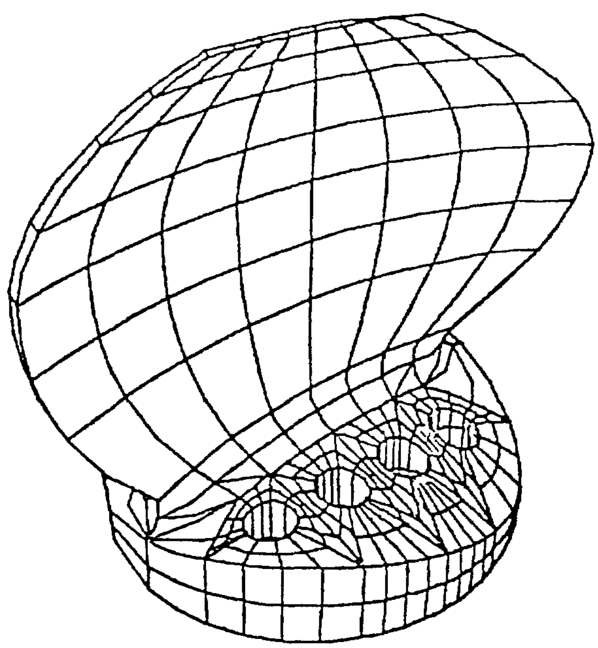


Figure 1.4: Blade-Fillet-Palm (Bolts as Superelements) Model

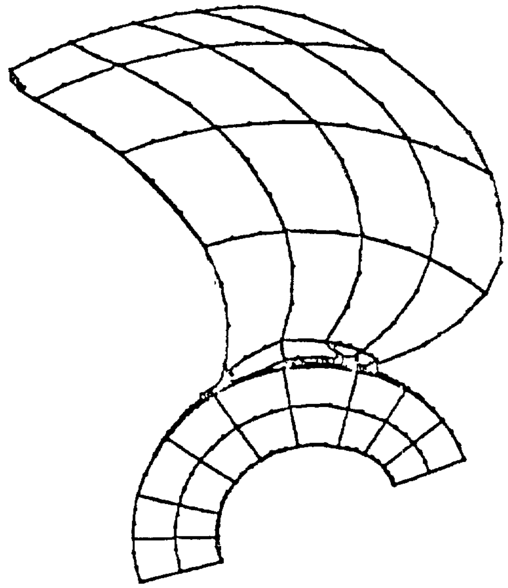


Figure 1.5: Blade-Fillet-Hub Model

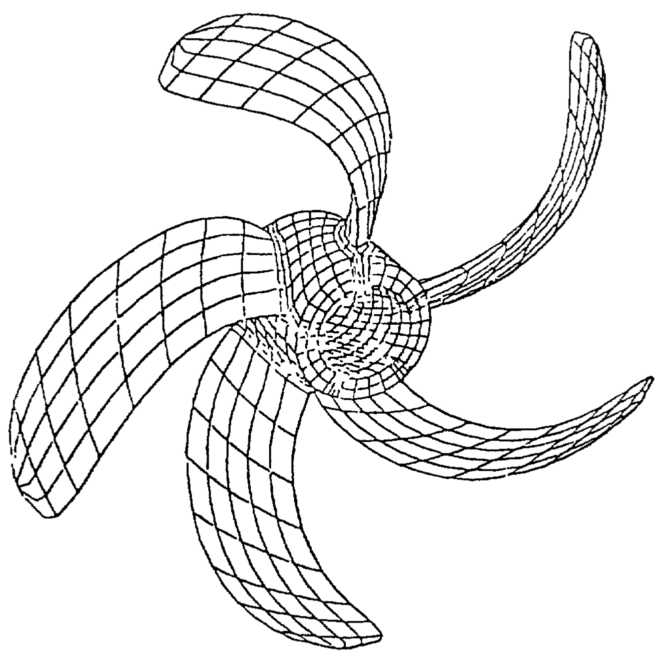


Figure 1.6: Multi-Blade Hub Model

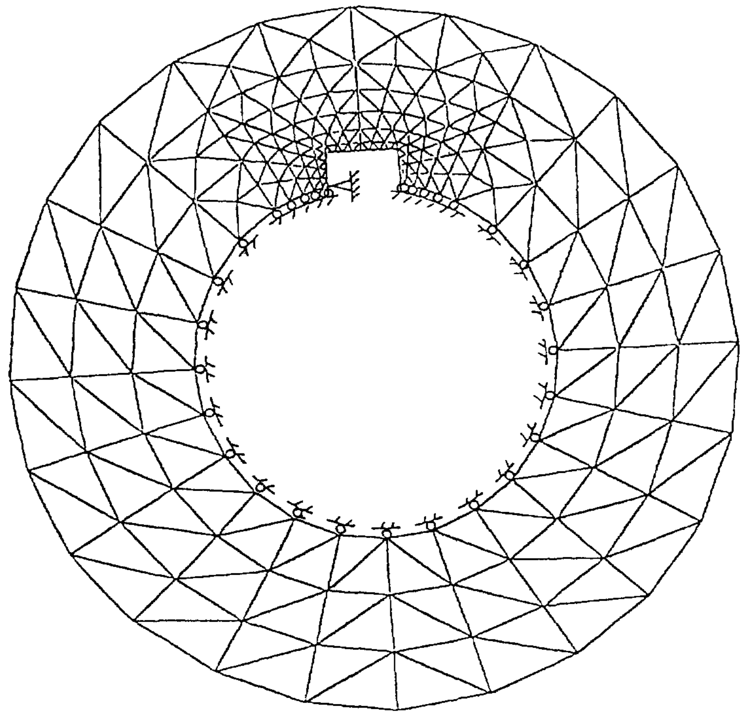


Figure 1.7: Keyway Model

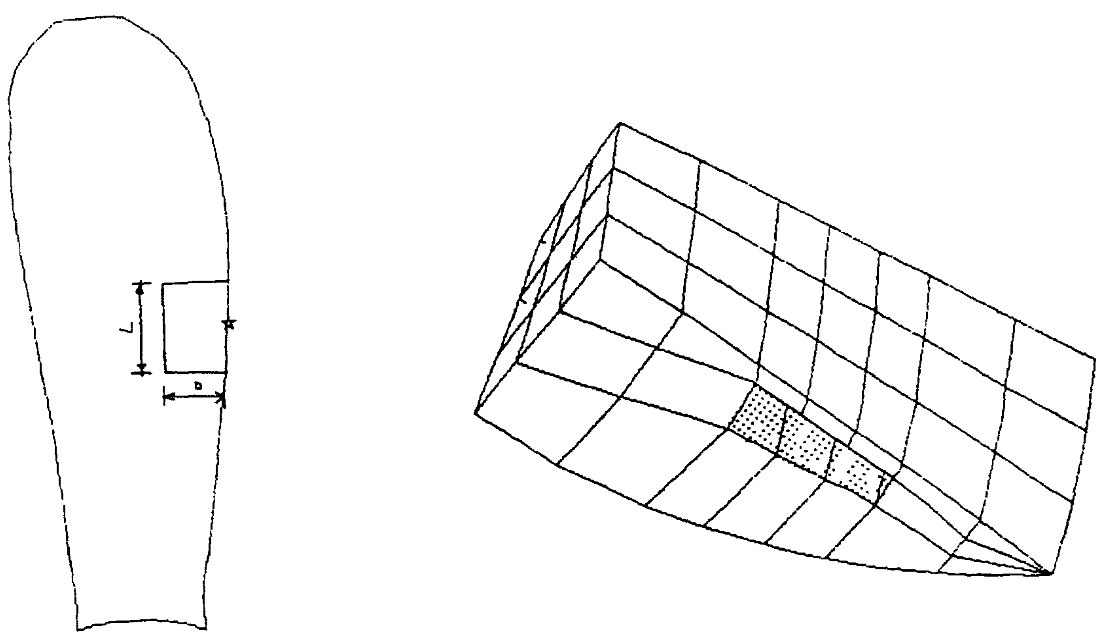


Figure 1.8: Air Channel Model

2. PROGRAM STRUCTURE

2.1 User Interface

PVAST is a menu driven program that is designed to be executed interactively. User interaction is through the use of a mouse and/or keyboard entries. It utilizes Menus, Sub-menus and Edit Tables for user response, as described below.

2.1.1 Main Menu

Figure 2.1 shows the PVAST main menu. The main menu has a single row of options which can be selected by clicking (using a mouse) on the appropriate option. This menu remains unchanged and any of the displayed options can be selected.

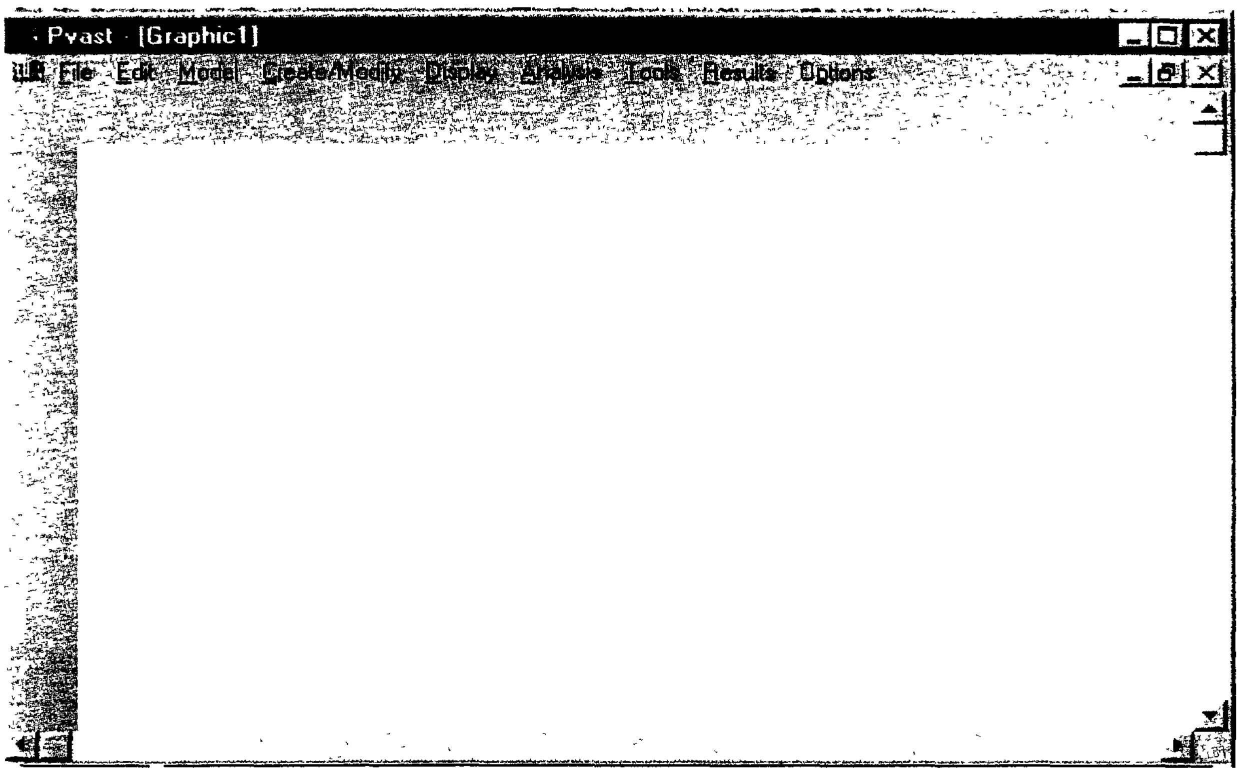


Figure 2.1: PVAST Main Menu

However, when an option to generate data or display models that are special to propellers (that is not general purpose), all the main menu options except **File** are temporarily deactivated (greyed out) and only the special purpose propeller options are activated. For instance, Figure 2.2 shows the screen that appears when the special purpose propeller blade data generation option is selected. As the figure shows, all the main menu options, except **File** have been deactivated and only the special purpose options are active at this time. On exiting the special purpose options, the original main menu (Figure 2.1) is restored.

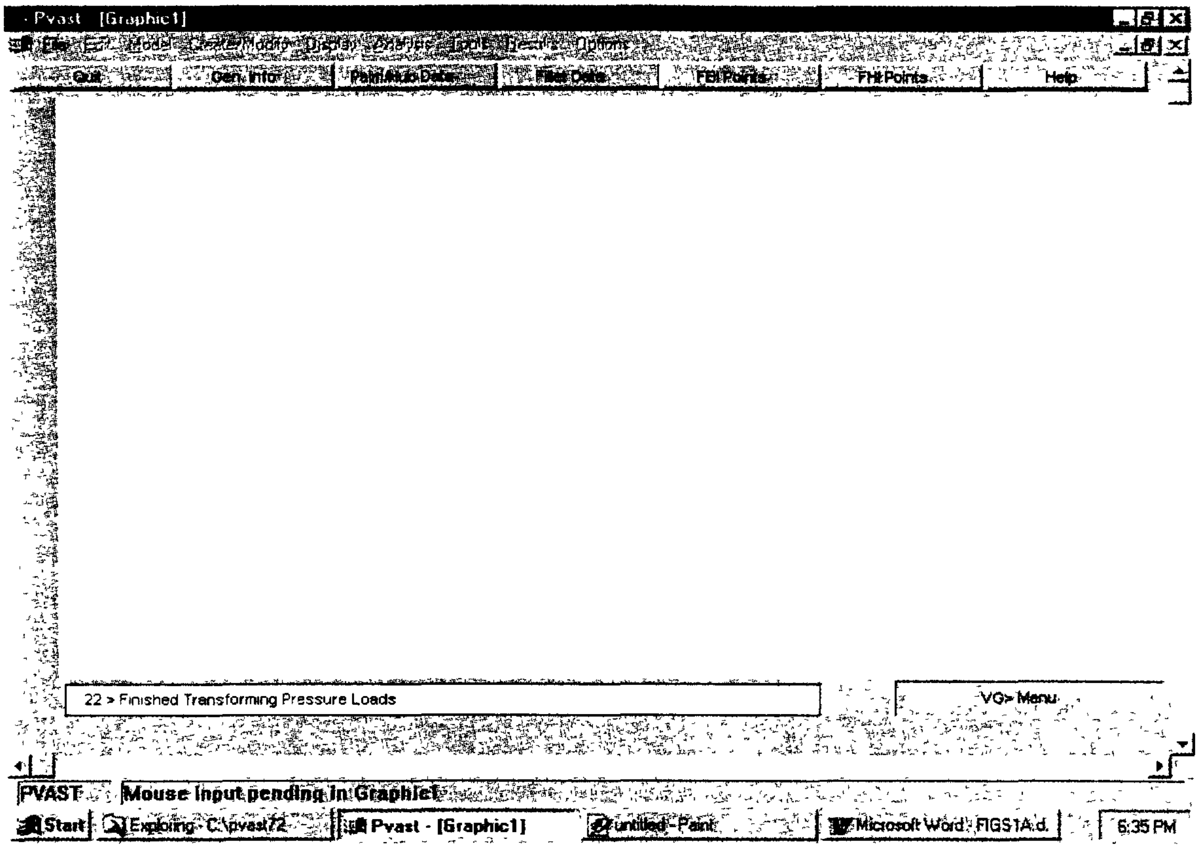


Figure 2.2: Main Menu for Special Propeller Modelling

Furthermore, when the general purpose finite element model display options are selected additional buttons associated with these modelling options are displayed as shown in Figure 2.3. Detailed descriptions of these buttons are provided in the MGDSA User's Manual [8].

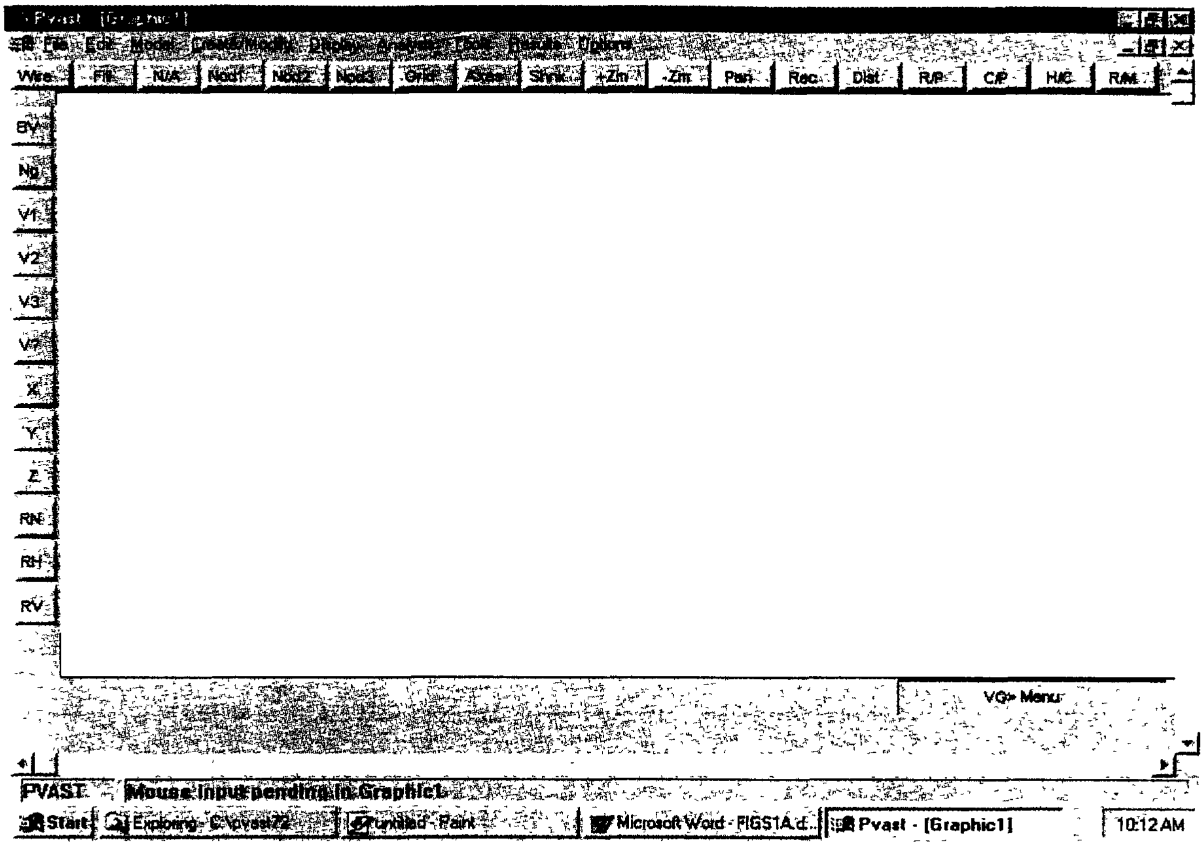


Figure 2.3: Additional Buttons

When an option on the main menu is selected, a submenu is displayed to enable further actions to be taken by the user. The PVAST submenu system is described in Section 2.1.2.

2.1.2 Submenus

The first level of submenus are activated by clicking on options on the main menu. Figure 2.4 shows a typical submenu obtained by clicking the **Create/Modify** option in the main menu. Depending on the submenu selected, second level submenus can be activated by clicking on options on the first level submenu and so on, until the final action sought is implemented. This final action could either be:

- A pick from a pick menu, or toggle switch;
- A keyboard entry via edit tables or an input box; or
- A plot that is displayed on the screen.

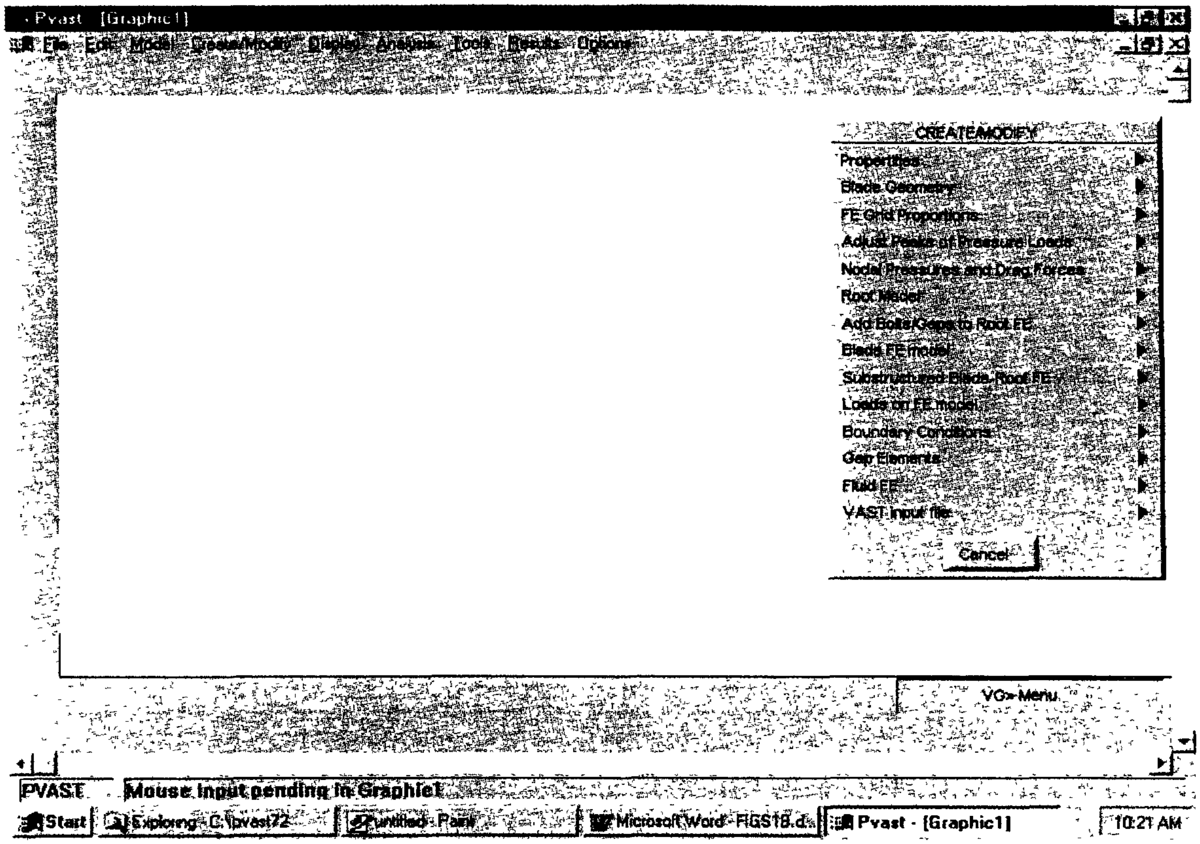


Figure 2.4: First Level Submenu

In general, the submenus are displayed on the right side of the screen (view port) as shown in Figure 2.4. However, the submenus via which edit tables are activated are displayed across the top of the view port as shown in Figure 2.2, because the edit tables tend to cover the whole view port. Recall that these types of submenus are associated with special propeller modelling or display options as described in Section 2.1.1. For this type of submenu, it is not possible to switch to the main menu (except to exit the program via the **File** option). For other submenus associated with the general purpose capabilities, it is possible to switch from the submenu to the main menu (see Reference [8]). Figure 2.5 shows samples of a pick menu, toggle switch and a submenu input box that can be used to implement the final action arising from submenus (see Reference [8] for further details). The Edit Table input dialogue boxes are described separately in Section 2.1.3.

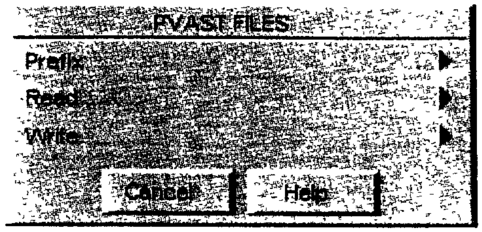


Figure 2.5(a): Pick Menu

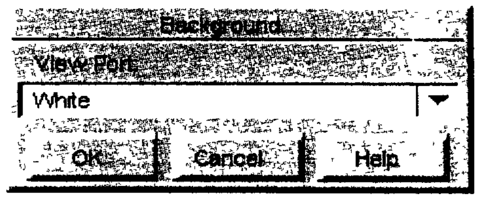


Figure 2.5(b): Toggle Switch

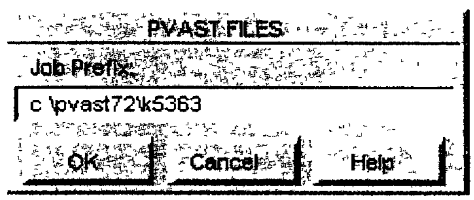


Figure 2.5(c): Submenu Input Box

2.1.3 Edit Tables

Edit tables are provided in PVAST to enable the entry of large amounts of data. This table replaces the terminal scrolling menus for entering and editing data in previous versions of PVAST. Figure 2.6 shows a typical edit table for entering/editing blade radial data. As shown, all of the radii can be defined by filling all of the boxes in this table, and all of the data is available for editing. Multiple Edit Tables can be used if the data does not fit on one table or page. The program automatically creates additional second pages if required, and the user can toggle between the pages by clicking on **PgUp** or **PgDn**.

BLADE DATA

NO	RADIUS	PITCH	CHORD	SKEW	RAKE
1	2 000E-01	6 068E+01	4 750E+02	-1 985E+02	-5 374E+01
2	2 500E-01	5 665E+01	4 910E+02	-1 695E+02	-6 718E+01
3	3 000E-01	5 316E+01	5 070E+02	-1 335E+02	-8 062E+01
4	4 000E-01	4.740E+01	5 350E+02	-4 252E+01	-1 075E+02
5	5 000E-01	4.240E+01	5 580E+02	8.400E+01	-1 344E+02
6	6 000E-01	3 755E+01	5 810E+02	2 425E+02	-1 612E+02
7	7 000E-01	3.236E+01	6 050E+02	4 365E+02	-1 881E+02
8	8 000E-01	2 686E+01	6 150E+02	6 605E+02	-2 150E+02
9	8 500E-01	2 444E+01	5 980E+02	7 900E+02	-2 284E+02
10	9.000E-01	2 246E+01	5 600E+02	9 200E+02	-2 419E+02

PgUp OK Cancel PgDn

Figure 2.6: Edit Table

2.1.4 Graphical Displays

PVAST provides options for displaying finite element models and input and the output data. The display options can be assessed by clicking the **Display** button on the main menu. In general, the graphical display options are provided through the MGDSA general purpose pre- and post-processing capabilities. However, special purpose plotting capabilities are also provided for displaying data and models that are specific to marine propeller structures, as discussed in Section 2.1.1. Figure 2.7 shows a plot of a blade finite element model using the general plotting capabilities, and Figure 2.8 shows a plot of blade radial data using the special purpose plotting option. By this approach all of the plotting capabilities available in earlier versions of PVAST are retained in this current version.

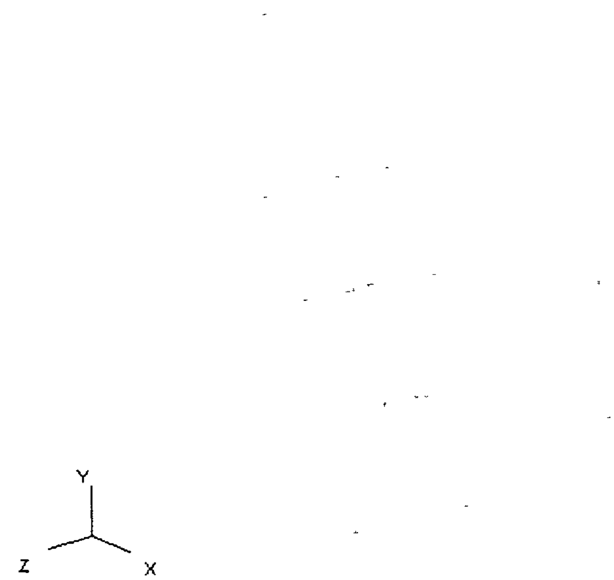


Figure 2.7: Plot of a Blade Finite Element Model

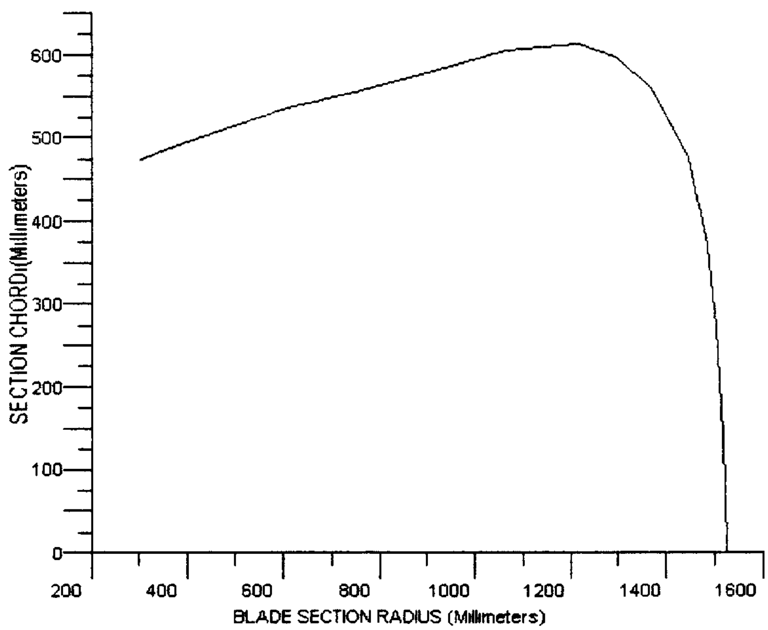


Figure 2.8: Blade Radial Data

2.2 File Structure

PVAST maintains a file structure through which the PVAST analysis data are stored, retrieved, and transferred among the various program modules.

The PVAST programs generally identify disk files as PREFX.XXX where the extension .XXX comprises of alpha and numeric characters. Note that extensions beginning with the letter S are associated with scratch files and those beginning with the letters L are associated with line printer output files. The PREFX is a unique alphanumeric file name prefix assigned to the current propeller analysis. The input data to PVAST modules is generally provided on the file .CDR that is generated when data for the various modules are generated or edited. Additional files are created after the creation of the various modules. These files may be required as input for plotting routines or for generation of subsequent models. Table 2.1 summarizes the input and output files for the various models and plotting options.

Table 2.1: Summary of I/O Disk Files for PVAST

Program Option	File Name Extension[1]		
	In	Out	Scratch
Creating and editing blade geometry	DAT	DAT,T16,PLT,LPB	
Plotting blade geometry	PLT		
Creating and editing blade FE grid properties	CDR,T16[2]	T11,L01[1]	
Plotting blade – root intersection	CDR,T11		
Creating and editing PVFIL fillet mode	CDR,T11	ROT,FIL,L02	
Creating and editing hub model PVHUB	ROT,FIL	HUB[5] ,L03	S61,S62,S63,S64
Creating and editing palm model PVPALM	ROT,FIL	PLM[5],T28,L04	
Creating blade FE model	CDR,T11,FIL[3]	T10[8],L05	
Creating sub-structure models	CDR,T10,T11,FIL,HUB,PLM,T28,T13,BHO	T27[9],L06,L07	
Creating fluid FE models	CDR,T11,CFM,T10	CFM,T30 or THN[11], L08, AMD	S61,S62,S63
Creating keyway models	CDR	T20[4],LOD	
Creating air channel models	CDR,T10,T11,LOD,T52	CHL	S61
Creating load models	LIN,LAD[10],CDR,T11,T12, T28	LAD,T12,L09,LOD,L10	
Boundary conditions data	TBC or (USE and T10,T20,T27 or PLM and BHO)	TBC,L11	S61,S62
Gap elements	GAP or (USE and T10,T20,T27 or PLM and BHO)	GAP,L11	S61.S62,S63
Bolt models	CDR[6], ROT, PLM[7], T13	T28,T13,BHO[12],L15	
Creating VAST input file PVUSE	T10,T27,T20,TBC,GAP	USE,CTL	
Plotting fluid FE model PGFLUD	T30		
Plotting displacements and stresses	T10,T11,T52,T53,T15,T18	T15,L31,T18,L32	
Plotting force vectors	T11,T41,T47,T14	T14,L33	
Computing force and moment resultants	CDR,T10,T11,T48,T52	TFR,L36	
Plotting mode shapes	T10,T11,T51		S61,S62

Notes:

1. Disk files are identified by names of the form PREFX.XXX where the extension generally comprises of one or two alpha characters followed by one or two numeric characters. Extensions beginning with the letter S refer to scratch files Those beginning with L refer to disk files intended for line printer output, copies of which can be obtained when user requests printing of files on the line printer All others are permanent files.
2. Optional (alternate source of blade data, instead of cards).
3. Optional (required when IBASE=1).
4. Contains geometry and superelement data for keyway model.
5. PREFX.GOM file for palm/hub analysis only.
6. Optional (alternate source of bold data, instead of through interactive prompts via terminal keyboard).
7. Original tape is replaced if palm model has been modified.
8. PREFX.GOM file for blade analysis only (no substructuring).
9. PREFX.GOM file for substructured blade and palm/hub analysis.
10. LAD file is used if available, otherwise LIN
11. Contains fluid element data for compressed (thin) fluid model.
12. Contains bolt and gap element data.

3. STEPS IN USING PVAST

3.1 Preliminaries

3.1.1 File Handling

The first action in using PVAST is to define a file prefix name to identify the current analysis session. This is done by clicking the **File** option on the main menu. The following submenu (Figure 3.1) appears:

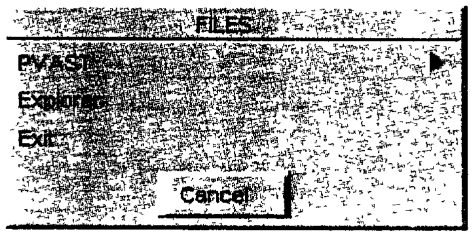


Figure 3.1: Files Submenu

Click on **PVAST** then **Prefix** and then enter the job prefix name, including the path, using the input box, as shown:

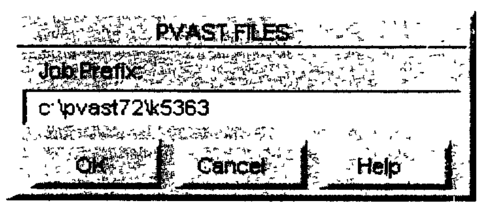


Figure 3.2: Files Input Box

The program will automatically append the appropriate three character file extension names to the file names. The file prefix entered could be a new or existing file. If it is an existing file, the program will use data in the database files for the various options, and the files will be updated as modifications are made to the existing data. If the file entered is a new one then data will be generated as the various steps are implemented.

For Windows 95/98/NT applications, the **Explorer** option on the **Files** submenu can be selected to browse the computer disk for files.

The **Exit** button on the **Files** submenu is used to exit the program entirely.

3.1.2 Edit

In general, the PVAST data are created and editing through the various Edit Tables, pick menus, switch toggles and data boxes. At the completion of each option, PVAST stores the data created onto the database files for subsequent retrieval. For some of the important text files, PVAST also provides the option to view or edit the data through standard Windows editors such as Notepad or Wordpad. These files include:

- The blade data file (.DAT)
- The card reader file (.CDR)
- The VAST input file (.USE)
- The VAST output file (.LPT)

Any other file may also be selected as shown in Figure 3.3. Again, the **Explorer** can be invoked to browse the disk for the appropriate file.

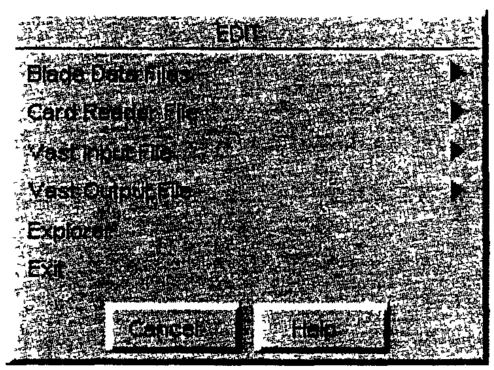


Figure 3.3: Edit Submenu

After the database file prefix for the current analysis has been identified, the user can now proceed with the various steps of the PVAST run, as described in Section 3.2.

3.2 Steps in PVAST Analysis

Although PVAST provides the user flexibility in the choice of modelling options it is desirable to follow a systematic order of steps, especially for new analysis sessions, to avoid undesirable effects due to wrong choice of analysis steps. In general, the program provides error or warning messages when options are selected in the wrong order. As the user becomes more familiar with the program, it would be possible to select options in any desired order, especially if database files exist.

However, the logical order of steps in any PVAST run is shown in Figure 3.4 by selecting the appropriate options (in boldface) on the main menu. The steps highlighted in Figure 3.4 are discussed below.

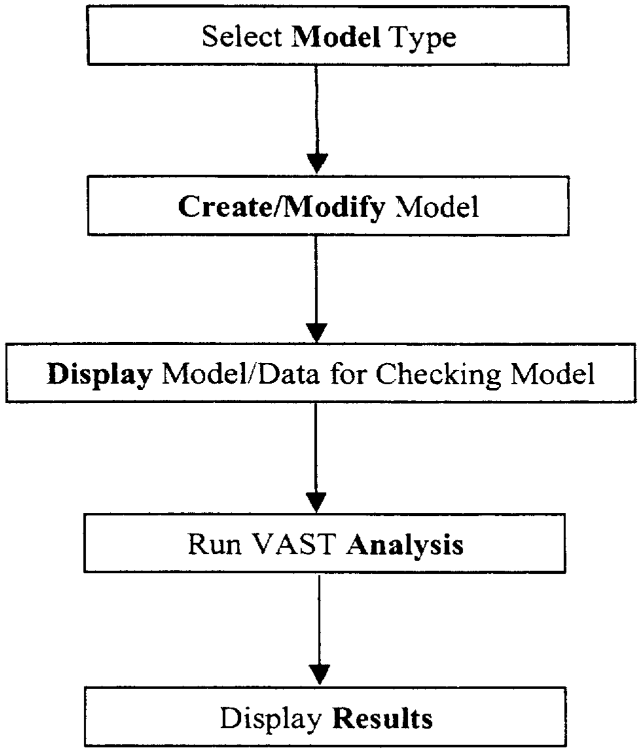


Figure 3.4: Steps in PVAST Analysis

3.2.1 Select Model Type

Figure 3.5 shows the **Model** submenu. The propeller model is selected by clicking on the appropriate model option which is then highlighted. If the model has already been selected, then the previously selected option will be highlighted on the **Model** submenu. Press **OK** when finished. The various types of models are shown in Figures 1.1 to 1.8.

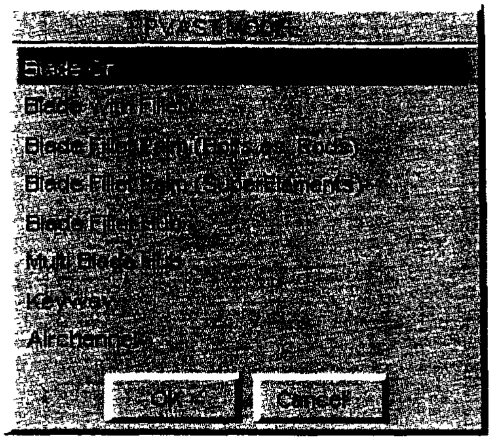


Figure 3.5: Model Submenu

3.2.2 Create/Modify

Figure 3.6 shows the **Create/Modify** submenu. There are a total of 14 options that can be selected from this submenu. However, not all of these options are required in any given run. The options required depend on the model, type of analysis, and whether the propeller is in air or water. Table 3.1 shows the options required for the various model and analysis types and fluid environment.

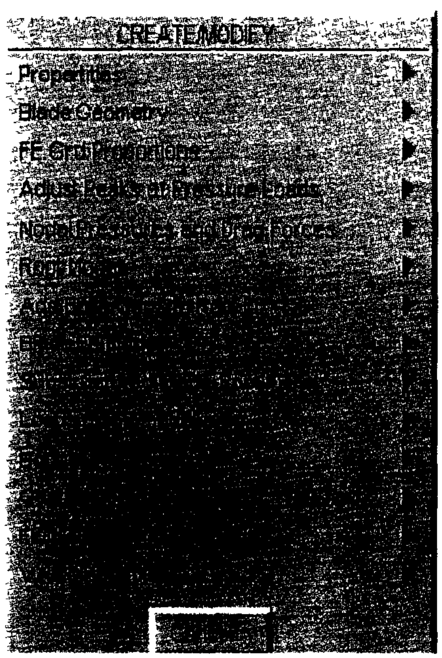


Figure 3.6: Create/Modify Submenu

The steps listed in Table 3.1 are applicable to all model types except the keyway and air channel models. The **Create/Modify** steps for keyway models are shown in Figure 3.7. It is desirable to access them in the order listed in the submenu.

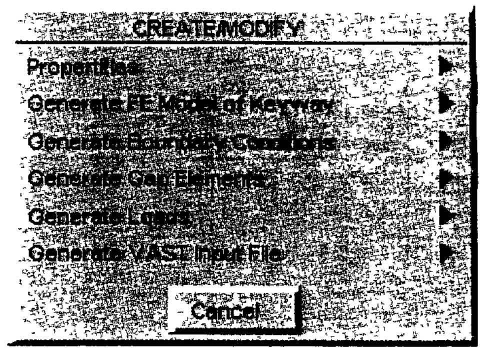


Figure 3.7: Keyway Create/Modify Submenu

For the air channel model, it is first necessary to do a complete analysis (from model generation to VAST analysis) of a blade-only model prior to the development of the air channel model using the top-down approach. Then the **Create/Modify** options for the air channel models are shown in Figure 3.8 and should be followed in the order shown.

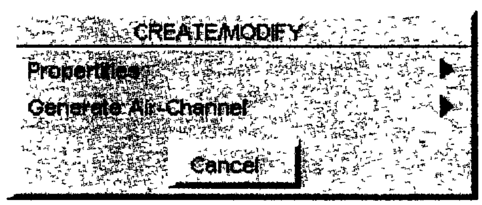


Figure 3.8: Air Channel Create/Modify Submenu

Table 3.1: Steps for Creating and Modifying Various Types of Propeller Blade Models

Create/Modify Options	Model Types					
	Blade Only	Blade with Fillet	Blade-Fillet Palm (Bolt as Rods)	Blade-Fillet Palm (Superelements)	Blade-Fillet Hub	Multi-Blade Hub
Properties	✓	✓	✓	✓	✓	✓
Blade Geometry	✓	✓	✓	✓	✓	✓
FE Grid Proportions	✓	✓	✓	✓	✓	✓
Adjust Peaks of Pressure Loads	✓[1]	✓[1]	✓[1]	✓[1]	✓[1]	✓[1]
Nodal Pressures and Drag Forces	✓[1]	✓[1]	✓[1]	✓[1]	✓[1]	✓[1]
Root Model		✓	✓	✓	✓	✓
Add Bolts/Gaps to Root FE			✓			
Blade FE Model	✓	✓	✓	✓	✓	✓
Substructured Blade-Root FE				✓	✓	✓
Loads on FE Model	✓[1]	✓[1]	✓[1]	✓[1]	✓[1]	✓[1]
Boundary Conditions	✓	✓	✓	✓	✓	✓
Gap Elements			✓			
Fluid FE	✓[2]	✓[2]	✓[2]	✓[2]	✓[2]	✓[2]
VAST Input File	✓	✓	✓	✓	✓	✓

1. Only if a static or transient analysis is required.
2. Only if structure is in water.

4. CREATING/MODIFYING PROPELLER BLADE GEOMETRY

4.1 Introduction

This chapter is concerned with the creation of the blade geometry. PVAST provides the capability to define and edit the data for generating the blade geometry. The capabilities described in this chapter are based on the BLADE program [2] developed by the Defence Research Establishment Atlantic (DREA). The program accepts the basic geometric data, converts these into three-dimensional spatial coordinates, and also provides a graphics capability for verifying the basic three-dimensional geometry and for producing conventional propeller blade drawings. In addition, it provides three-dimensional coordinates in tabular form. It also generates a data file of basic geometry from the data entered. All other geometric data, including geometry plotting and geometric data files, are generated from this file, The program allows editing of this basic file at any stage of data entry, and also allows the entry of data to be interrupted and restarted later without any loss of the existing data. Help statements can be displayed at the user's request at any data entry prompt. The program creates a data file for use in generating the finite element models described in Chapter 5. Analytical models of the propeller blade, and the blade plus hub or palm, can be generated from these data.

It is essential that the user be familiar with the definitions of certain blade geometric parameters in order to be able to define the blade geometry effectively. To this end, detailed descriptions/definitions of the parameters are provided in the following subsections. A description of the geometry of marine propellers is provided in Reference [5].

To define the blade geometry, select **Create/Modify** from the main menu, then **Blade Geometry**. Then the blade geometry menu appears as presented in Figure 4.1, showing the steps in the definition of the blade data. The steps involved are:

- Definition of **Basic Data**
- Definition of **Radial** data
- Definition of **Chordwise** data
- Definition of **Edge** data
- Definition of base (**Palm/Hub**) data

The first step (definition of basic blade data) must be defined first, and the other steps may then be defined in any desired order. Detailed descriptions of the steps are presented in Sections 4.2 to 4.6.

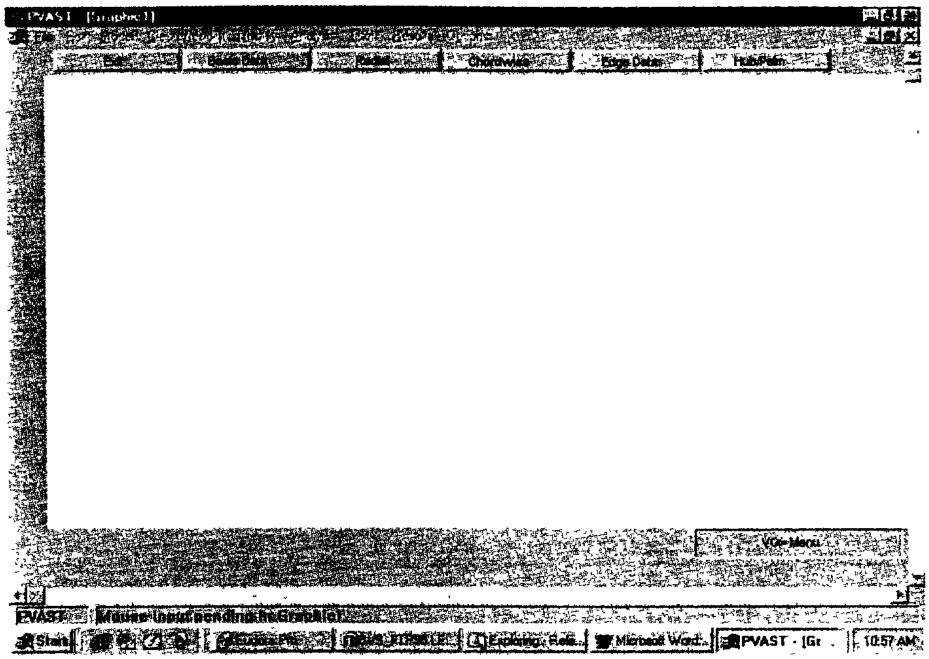


Figure 4.1: Blade Geometry Menu

4.2 Basic Blade Data

Figure 4.2 shows the **Basic Data** submenu. Descriptions of the terms/parameters are provided below.

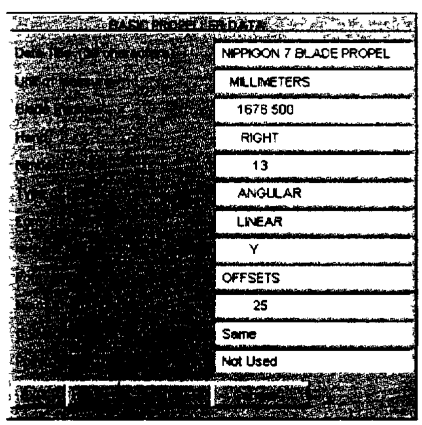


Figure 4.2: Basic Blade Data Submenu

Data Title A 39 character string used to identify the entered data.

Unit of Measure The data may be entered in units of millimeters or inches. There is a conversion option, in the Blade Geometry Processor that may be used if the geometry is not in the units required for the analysis.

Blade Radius Meaning the full blade radius, is the dimension from the center of rotation to the tip of the blade.

Hand The term used to describe the direction of rotation of the propeller.
Right hand rotation, +1 is clockwise (Looking Forward)
Left hand rotation, -1 is counter clockwise (Looking Forward)
The propeller axis coordinate system is shown in Figure 4.3.

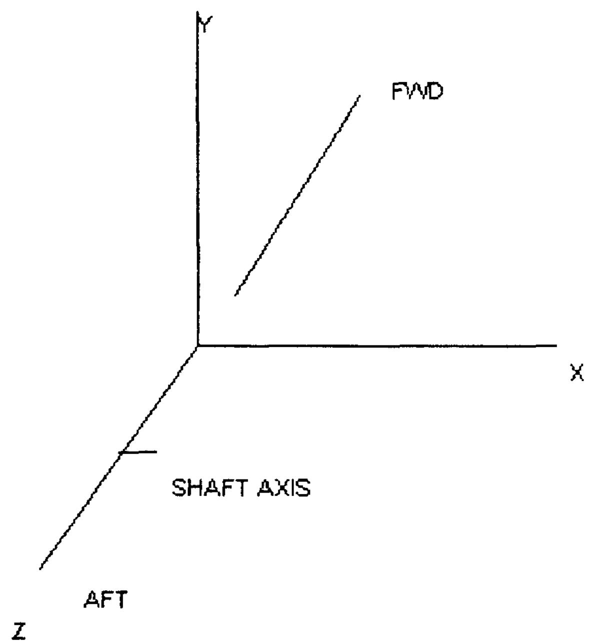


Figure 4.3: Propeller Axis Coordinate System

Number of Radii The blade is defined by a number of blade sections, which are located at specified radial distances from the center of rotation. A maximum of 31 sections can be specified. If enough sections have not been specified in the region of the blade tip to properly define it then it will be possible to define it later. For this situation the number of sections entered should be reduced from 31 by an amount which will allow sufficient new sections to be generated to define the tip.

Type of Pitch

There is an option to describe the setting of each prescribed blade section relative to the blade datum either as:

- An angle (the angle between the chordline of the section at radius r and a plane normal to the shaft axis);
- As pitch (the distance traveled along the shaft axis in one revolution); and
- As the pitch diameter ratio, advance of the section per revolution divided by the radius, for each radius.

Form of Skew

Skew can be specified in two forms:

- Skew Angle, the angular displacement in the plane of rotation about the shaft axis from the vertical axis of the blade by the blade section reference point, generally midchord.; and
- Linear Skew, a linear dimension that is the displacement of the blade section along the developed pitch helix measured from the origin of the vertical axis.

For a right hand blade positive skew angles are counter-clockwise, and positive linear skew is in the X direction.

Is there Rake

Rake is the displacement of the section reference point (generally the mid chord point) fore or aft of the propeller rotation plane. Positive rake is an aft displacement.

There are three types of rake:

- Imposed rake a specified rake;
- Skew-induced rake, produced by the skew; and
- Net rake, the desired rake of the blade.

The program requires the net rake. This is obtained by adding the skew induced rake ($SKR_i = SK_i Sin\phi_i$) to the imposed rake, where SKR_i, SK_i, ϕ_i are the skew induced rake, skew, and pitch angle at the blade section i, respectively.

Form of Data

There are three forms of data that the program can process:

- *Non-dimensional ratios* of max thickness, to chord length, plus ratios of max camber to chord length for each section. The sections are defined by ratios of basic section half thickness to max thickness for the basic section at specified chord fractions together with ratios of camber to max camber;

- *NACA section data* (see Abbott and von Doenhoff, camber. Appendices 1 and 2); and
- *Section offsets* of blade sections presented in tables or on drawings.

The offsets may have the midchord or point of max thickness as the datum.

No. of Chord Points The maximum number of chord stations allowed is 40. Often there are not enough stations in the vicinity of the leading and trailing edges to adequately define them. In such a case it is prudent to reduce the number of stations specified at this time to 36 or less. This will allow additional stations in these regions to be generated later in the input.

Common Chord Fractions The option to define the chordwise stations for each radial position is available. Common station locations need only be entered for the first radial position.

Point of Max Thickness (PMT) OFFSETS or Tabular data may use leading edge or the point of maximum thickness, PMT as the section chord fraction datum. If leading edge is used, then the chord stations are input as fractions of the full chord.

If the point of maximum thickness is used, then chord stations are input as fractions of chord length to the PMT and as fractions of the chord length beyond the PMT. Data input order is from the leading edge to the trailing edge, with leading edge=1.0, point of maximum thickness=0.0, and trailing edge=1.0 If the point of maximum thickness has been specified as the section datum then its location must be specified for each section as a fraction of the full chord length.

The **Basic Blade Data** table is self-explanatory. However, a **Help** option is provided to explain the input parameters. The **Help** also includes a plot option, which defines the propeller axis coordinate system as shown in Figure 4.3. Default values are assigned wherever possible. These may be changed by mouse clicks, which toggles them to other definitions/parameters. On completion of the table click **OK** returns to the **Blade Geometry** menu.

4.3 Radial Data

RADIAL BLADE DATA IN MILLIMETERS AND DEGREES				
RADIUS	CHORD	PITCH	SKEW	RAKE
1760	671 9410	67 0710	0000	0000
2000	678 3120	67 0530	- 1720	8290
2500	769 5130	66.2530	-2 3940	12 2500
3000	857 0270	63 3490	-1 3450	20 3150
3500	942 5280	59 8530	3 2710	25 6720
4000	1025 0120	56 1820	11 8080	28 0790
5000	1200 0000	48 9740	42 3980	24 6800
6000	1339 8590	42 2670	93 2480	12 7250
7000	1375 0650	36 0710	168 5650	-5 3650
8000	1308 0050	30 1340	272 6280	-24 2060
9000	1095 0900	24 2800	410 0080	-37 1540
9500	829 8670	21 2960	492 6480	-38 0950
9750	564 9810	19 7240	537 2630	-36 1360

OK Cancel HELP

Figure 4.4: Radial Data for Definition of Blade Geometry

Figure 4.4 shows the edit table for defining **Radial Data.** Descriptions of the terms/parameters are provided below.

Radius Blade section locations must be entered as fractions of the full blade radius. eg. 0.50 is half the full radius. Locations must be in ascending order.

Chord Chord length is defined as the distance from the leading to the trailing edge for a fully developed blade section. It passes through or nearly through the fore and aft ends of the section. The chord lengths at each blade section is required.

Pitch The Pitch is described as the setting of each blade section relative to the blade datum, determined in the Basic Data table. The pitch (defined as an angle, pitch or pitch diameter ratio, as described under Basic Blade Data) is required at each radial section.

Skew Skew is specified in either of two forms, determined in the Basic Data table. The skew, defined either as skew angle or linear skew, as described under Basic Blade Data is required at each radial section.

Rake

Rake is the displacement of the section reference point (generally the mid chord point) fore or aft of the propeller rotation plane. Positive rake is an aft displacement.

There are three types of rake determined in the Basic Data table:

- *Imposed rake* is a specified rake.
- *Skew-induced rake* is produced by the skew.
- *Net rake* the desired rake of the blade.

The program requires the net rake. This is obtained by adding the skew induced rake ($SKR_i = SK_i Sin\phi_i$) to the imposed rake, where SKR_i, Sk_i, ϕ_i are the skew induced rake, skew, and pitch angle at the blade section i, respectively.

If **Help** is required it should be selected prior to data entry, because once data entry into the table is started it must be completed before a selection can be made. Selection of a **Help** topic presents the appropriate help message. To return to the **Radial Data** pasteboard, click the **Cancel** button.

The **Radial Data** table is common to all Forms of Data, the only difference being in the units of pitch and skew as defined above. Once the table is filled, the forced entry is no longer operational. At this point the data can be edited at random and **Help** may be selected at any time. To return to the **Radial Data** pasteboard click **OK**. The plot options may now be used (as described in Chapter 6) to graphically check the data. Click **Cancel** to return to the Blade Geometry main menu.

4.4 Chordwise Data

The **Chordwise Data** command button will present pasteboards and tables that are determined by the **Form of Data** selected in the **Basic Data** table. As noted in Section 4.2, the forms of data include Offsets, Non-dimensional ratios of maximum thickness and NACA section data. The data required under these three types are described below.

4.4.1 Form of Data: Offsets

CROSS SECTION AT RADIUS 175					
CHORD FRACTION	BACK OFFSET	FACE OFFSET	CHORD FRACTION	BACK OFFSET	FACE OFFSET
0000	0000	0000	5000	-66 1530	57 8700
0075	-10 4760	9 8400	5500	-64 4930	56 9800
0100	-12 5180	11 6900	6000	-61 8120	55 0700
0250	-20 7650	18 8900	6500	-58 0830	52 0300
0500	-29 9280	26 5000	7000	-53 3990	47 8800
1000	-43 0650	37 1200	7500	-47 7620	42 6900
1500	-51 9070	44 1200	8000	-41 1630	36 5000
2000	-57 3440	48 2700	8500	-33 6040	29 4000
2500	-61 5360	51 7500	9000	-24 8620	21 4000
3000	-64 4190	54 3400	9500	-15 0920	12 7600
3500	-66 1590	56 1500	9750	-9 4610	8 2200
4000	-66 9590	57 3400	1 0000	-4 1590	4 1600
4500	-66 9520	57 9500			

RADII OK Cancel HELP

Figure 4.5: Chordwise Data for Defining Blade Geometry

Figure 4.5 shows the edit table for defining **Chordwise Data** when the form of data is Offsets. Descriptions of the terms/parameters are provided below.

Chord Fraction Each chord station location must be specified as a fraction of the full chord length. For example a station 20 mm from the leading edge in a chord length of 100 mm is entered as 0.2

If the point of max thickness is used, (determined from the **Basic Data** table), then the chord stations are input as fractions of chord length to the PMT and as fractions of the chord length beyond the PMT. The order of data input is from the leading edge to the trailing edge, with the leading edge=1.0, point of maximum thickness=0.0, and trailing edge=1.0

Back Offsets The back offset defines the forward facing surface of the propeller blade. It is the dimension from the chord line, at the specified chord station, of the section, to the back of the section. Offsets are negative above the section chord line.

Face Offsets The face offset defines the aft facing surface of the propeller blade. It is the dimension from the chord line, at the specified chord station, of the section, to the face of the section. Offsets are positive below the chord line and negative above.

It is recommended that the options available in this section be selected in sequence. The **Help** option provides definitions of the parameters. Note if **Help** is required for the tabular data it should be chosen before editing the data because the forced data entry will make it unavailable until after the table is filled. During initial input the entry table for a given cross-section is a forced entry until it is filled. Following that, random editing is then available, as well as **Help** and the other command buttons. Selecting **OK** will advance entry to the next radius table. Once all the data is entered the **RADI** button is used to access any radial location in the data block. Graphic plots are available to assist in checking this data.

4.4.2 Form of Data: Ratios

The **Basic Data** input and the **Radial Data** input are common to all forms of the data input, with only two exceptions. In the **Basic Propeller Data** table in Figure 4.2, the **Form of Data** is set to **Ratios**, and **Point of Max Thickness** is set to **Not Used.**

Figure 4.6 shows the pasteboard for defining the **Chordwise Data** when the form of data is **Ratios**. Descriptions of the parameters are provided below.

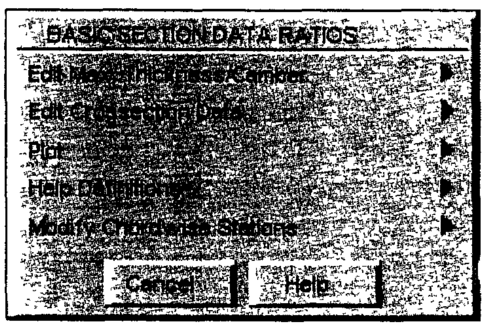


Figure 4.6: Chordwise Data When Form of Data is Ratios

Edit Max Thickness/Camber Clicking on this option presents an edit table, as shown in Figure 4.7, for defining the values of maximum thickness and maximum camber as fractions of chord length for each radial location. Enter the appropriate values. Click **OK** when finished, and return to the **Basic Section Data Ratios** pasteboard.

BASIC SECTION DATA RATIOS INCHES

RADIUS	Max. Thickness Fractions	Max. Camber Fractions
[illegible]	.103	037
[illegible]	.027	033
[illegible]	016	027

OK Cancel HELP

Figure 4.7: Maximum Thickness/Camber Defined as Ratios

Edit Cross Section Data

Clicking on this option presents an edit table for defining the cross section data as shown in Figure 4.8. The Chord stations are entered as fractions of the chord length. **Half Thickness** is expressed as a fraction of the **Maximum Thickness** at that radial position, defined in the previous table (Figure 4.7). Camber Offset is expressed as a fraction of the **Maximum Camber** at that radial position defined in the previous table (Figure 4.7).

EDIT RATIOS SECTION DATA

Fraction Chord	Half Thickness Fraction of Max.	Camber Offset Fraction of Max.	Fraction Chord	Half Thickness Fraction of Max.	Camber Offset Fraction of Max.
0000	0000	0000	4500	5000	9880
0120	1160	0910	5000	4960	1 0000
0250	1530	1590	6000	4650	9780
0500	2090	2710	7000	4030	8890
0750	2540	3660	8000	3110	7030
1000	2920	4480	9000	1880	3590
2000	4000	6990	9500	1140	1710
3000	4640	8630	1 0000	0330	0000
4000	4950	9610			

OK Cancel HELP

Figure 4.8: Cross Section Data in Form of Ratios

4.4.3 Form of Data: NACA

The **Basic Data** input and the **Radial Data** input are common to all forms of the data input, with only two exceptions. In the **Basic Propeller Data** table, Figure 4.2, **the Form of Data** is set to **NACA**, and **Point of Max Thickness** is set to **Not Used**.

Figure 4.9 shows the pasteboard for defining the **Chordwise Data** when the form of data is **NACA.**

Descriptions of the parameters are provided below.

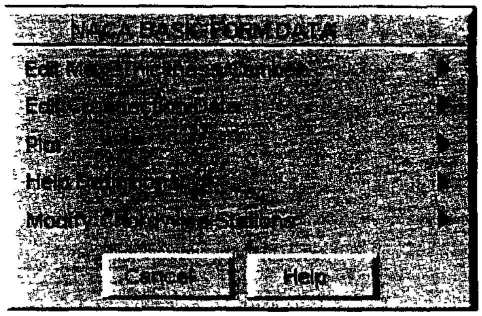


Figure 4.9: Chordwise Data When Form of Data is NACA

Edit Maximum Thickness/Camber

Clicking this option presents a table, shown in Figure 4.10, for defining the values of maximum thickness and maximum camber as a fraction of chord length for each radial location. Enter the appropriate values. Click OK when finished and return to the **Basic Section Data NACA** pasteboard.

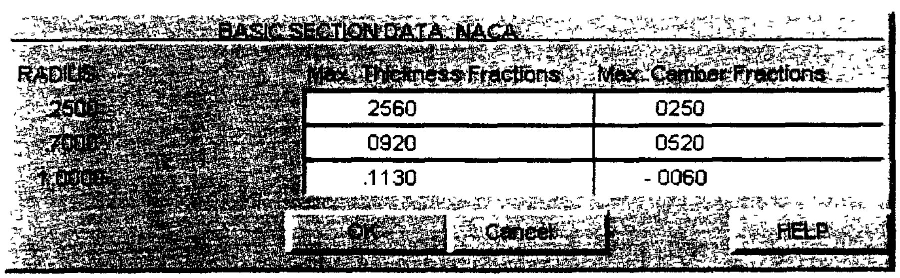


Figure 4.10: Maximum Thickness/Camber Defined as NACA Data

Edit Cross Section Data

Clicking on this option presents the edit table shown in Figure 4.11, for defining the NACA cross section data.

EDIT NACA BASIC FORM DATA

% CHORD	HALF THICKNESS %CHORD	CAMBER OFFSET %CHORD	% CHORD	HALF THICKNESS %CHORD	CAMBER OFFSET %CHORD
0000	0000	0000	40 0000	5 8850	5 7600
1 2500	1.2920	2960	50 0000	6 0000	6 0000
2 5000	1 8050	5850	60 0000	5 8350	5 7600
5 0000	2 5090	1 1400	70 0000	5 2690	5 0500
7 5000	3 0320	1 6650	80 0000	4 1990	3 8400
10 0000	3 4570	2 1600	90 0000	2 5170	2 1000
15 0000	4 1350	3 0600	95 0000	1 4150	1 1400
20 0000	4 6640	3 8400	100 0000	1 2000	0000
30.0000	5 4170	5 0400			

OK Cancel HELP

Figure 4.11: NACA Cross Section Data

The Chord stations are entered as percent of the chord length.
Half Thickness is expressed as a percent of the chord length.
Camber Offset is expressed as a percent of the chord length.
These values are of a typical section. The previous table establishes a ratio between these values and the section location being generated. Based on the maximum thickness, or camber of the typical section and those values in the table in Figure 4.12.

BASIC SECTION DATA NACA

RADIUS	Max. Thickness Fractions	Max. Camber Fractions
.2500	2560	0250
.7000	0920	0520
1.0000	1130	- 0060

OK Cancel HELP

Figure 4.12: Maximum Thickness/Camber Defined as NACA Data

4.5 Edge Data

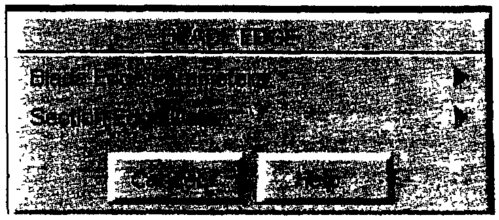


Figure 4.13: Edge Data for Defining Blade Geometry

Figure 4.13 shows the edit table for defining **Edge Data.** The options should be accessed in order.

Blade Edge Parameters

The table defines all the single variable terms relating to blade edge generation. The definition of Coordinate Print Option for the printer file has been added here for entry convenience. HELP definitions for all variables are accessible at all times.

Section Edge Data

The Leading and Trailing Edge Data Table, displays the radial locations of each section for reference only, they may not be changed. All variables should be defined, they may be set to 0.0 or a positive value.

Help definitions are available.

Descriptions of the terms/parameters are provided below.

Blade Edge Parameters

Leading Edge Points
The opportunity exists at this stage to improve the shape and smoothness of the leading edge by specifying additional stations up to the maximum indicated less any required for trailing edge enhancement.

Trailing Edge Points
The opportunity exists at this stage to improve the shape and smoothness of the trailing edge by specifying additional stations up to the maximum indicated. A zero entry will cause any trailing edge enhancement to be ignored.

Edge Type

There are four options for leading and trailing edge generation beyond that of the raw data provided. One causes an edge ending with a previously specified radius used. The second generates an elliptical edge for each section. The third uses the radial method up to the section specified then the elliptical method is used for the remaining sections. The fourth uses the elliptical method up to the section specified by a negative section number, then the radial method is used for the remaining sections.

Number of Blade Tip Radii

The blade tip is often left incomplete. If this is the case, additional data can be supplied at this stage which will allow the tip to be generated. A number of additional radii beyond the last radius fraction must be specified. A zero entry will end the blade tip at the last radius given in the original data. Total number of radial sections 30.

Blade Tip Pitch

A pitch is specified at the blade tip in the same units as the pitch was entered, defined in **Basic Blade Data,** for the radial sections. If it is not available the program uses an estimated value.

Blade Tip Skew

If skew is specified for the radial sections it must also be given for the blade tip in the same units as for the radial sections. Form of Skew is defined in **Basic Blade Data.**

Blade Tip Rake

If rake is specified for the radial sections it must also be given for the blade tip.

Blade Tip Point of Max. Thickness

If point of maximum thickness is specified for the radial sections it must also be given for the blade tip.

Blade Tip Generation Method

There are two methods of blade tip profile generation: *Radial* and *Elliptical.* The *Radial* method uses a blade tip radius blended into the cross-section shape.

The *Elliptical* method uses a blade tip elliptical semi-thickness blended into the cross-section shape.

Blade Tip Radius

The radius or elliptical semi thickness that is to be used to generate the blade tip thickness.

Distance to Center of Blade Tip

The drawing of the section showing the maximum thickness will show graphically the distance of the blade tip profile CT from the vertical reference axis. If a dimension is not known and the blade tip and the vertical axis do not coincide it can be estimated by the program by linear interpolation, otherwise it will be necessary to estimate it by trial and error. Dimension CT is without any skew or rake that may be present, and is negative forward of the reference axis and positive aft. The maximum thickness diagram is drawn without rake or skew therefore the tip profile must be generated without them.

Coordinate Print Option

Printer file options. File PREFX.LPB

- No detailed blade coordinate listings are produced.
- Listings of x,y,z Cartesian coordinates of the blade together with the same data in the form of Cylindrical coordinates is produced.
- Listing of coordinates suitable for blade measurement is produced. These provide for back and face measurements by supplying the coordinates of the face surface as well as the back, after rotating the coordinates through 180°. Listing of coordinates in a form that is compatible with a measurement machine.

Rotation Angle for Measurement

It may be necessary to remove some of the effect of the pitch of the blade to facilitate measurement.

Shim Thickness for Measurement

The location of the prop center of rotation and the center of rotation for measurement may be different. A positive or negative adjustment may be used.

Leading and Trailing Edge Data

LEADING AND TRAILING EDGE DATA

RADIUS	Leading Edge Radius	Trailing Edge Radius	Antisinging Edge Angle	Antisinging Edge Length
1760	17 1345	6 3162	0000	0000
2000	16.8221	6 3083	0000	0000
2500	12 3892	6 0022	0000	0000
3000	9 2559	5 6564	0000	0000
3500	6 8805	5 3724	0000	0000
4000	5 2276	5 0226	0000	0000
5000	3 0001	4 5601	0000	0000
6000	1 8758	4 0196	0000	0000
7000	1 2376	3 5752	0000	0000
8000	9156	3 1392	0000	0000
9000	5475	2 7377	0000	0000
9500	4149	2 4896	0000	0000
9750	3390	2 3729	0000	0000

HELP OK Cancel

Figure 4.14: Edit Table for Defining Leading Edge Radius

Figure 4.14 shows the edit table for defining **Leading Edge Radius.** Descriptions of the terms/ parameters are provided below.

Leading Edge Radius

A leading edge radius should be prescribed if a smooth leading edge is to be generated. A zero entry is acceptable. It will cause only the specified section data to be used, which may result in a sharp point at the extreme edge.

Trailing Edge Radius

A trailing edge radius should be prescribed if a smooth trailing edge is to be generated. A zero entry is acceptable. It will cause only the specified section data to be used, which may result in a sharp point at the extreme edge.

Anti-singing Edge Angle

An anti-singing trailing edge may be defined for any blade cross-section. A zero entry results in no change to the trailing edge. It is defined as an angle and a length. The angle, in degrees, is added to the slope of the back surface of the cross section and the length, in the units of measure, parallel to the baseline.

Anti-singing Edge Length Used to specify the anti-singing edge length, which together with the anti-singing edge angle described above, defines the anti-singing trailing edge.

4.6 Hub/Palm Data

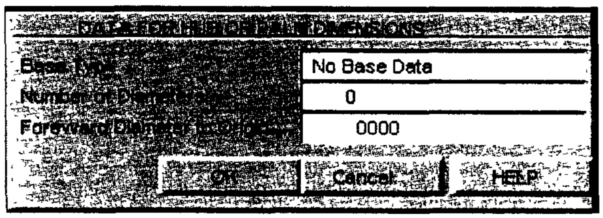


Figure 4.15: Edit Table for Hub/Palm Data

Figure 4.8 shows the edit table for defining the base **(Hub/Palm Data).** Descriptions of the terms/parameters are provided below.

Base Type The blade is attached to a hub or, in the case of a variable pitch propeller, to a palm. If data are available for describing the attachment it can be entered at this point in the program. Three options are given: *No Base Data*, *Hub Data*, or *Palm Data*.
The data entered are available for later plotting of the hub or palm shape as a check of the first given blade section location. This check will confirm the hub or palm and blade data relationship

For *No Base Data*, no further data entry required.

Number of Diameters The other two options require a description of the hub shape. The hub of the propeller is described by a minimum of two diameters. Complex hub shapes may use up to 6 diameters to properly describe them. When more than 2 diameters are given a spine interpolation is used to generate additional diameters for drawing the hub outline.

Forward Diameter to Origin The distance measured from the forward end of the hub to the blade data origin is required to locate the blade sections relative to the hub outline.

Palm option:

Palm Diameter The palm is the base to which the blade of a variable pitch propeller is attached. It is circular so that it can rotate with the blade to change pitch. The diameter of the palm is required as input at this point.

Spherical Palm Radius A variable pitch propeller may be mounted on a spherical portion of a hub the palm profile requires this radius.

Palm Base Height The Palm Base Height is the distance from the shaft center to the bottom of the palm.

Palm Coordinate Offset The Palm Coordinate Offset is the distance off the center of the shaft to the center of the Spherical Palm Radius.

Hub Option:

Hub Diameters The diameters describing the hub outline are required at this point. These are entered with the most forward diameter first, at location = 0.0.

4.7 Blade Geometry Data Files

The data files created by the blade geometry generation module are permanent files, which will be retained unless deliberately deleted. They are:

PREFX.DAT - basic data file
PREFX.LPB - tabular geometric data file
PREFX.PLT - plotting file
PREFX.T11 - required for post processing
PREFX.T16 - data file for PVAST

Once the file PREFX.DAT has been created, all the other files can be created from it. If it is necessary to reduce file storage requirements to an absolute minimum, only PREFX.DAT need be retained. A typical example of a PREFX.DAT file is shown in Appendix A.

The PREFX.LPB file, depending on the coordinate print option chosen, contains geometric data useful for manufacture and checking which can be hard copied to the printer.

PREFX.PLT contains the geometric data required for plotting. This file must exist if any plotting of three-dimensional data is to be done. If it does not exist or if changes have been made to the PREFX.DAT file, then the blade generation option from the starting menu must be chosen.

PREFX.T16 is a data file for generating the propeller finite element models in the subsequent steps in program PVAST. With this file, PVAST can be used to create finite element models of the blade alone, the blade in conjunction with the hub or palm, or a complete monobloc multi-bladed propeller.

The PREFX.PLT file is used to produce plots of all the basic data plus the additional coordinates generated from the leading edges, and the blade tip. The following drawings can be made, as discussed in Chapter 6.

(a) final data plots
(b) plots of individual developed blade sections
(c) blade thickness plots
(d) transverse view of the blade
(e) plan view of the blade
(f) longitudinal view of the blade
(g) pitch diagram
(h) plan view of stacked blade sections
(i) plan view of first blade section and blade outline
(j) outline of area swept by the blade
(k) combined geometry drawing
(l) expanded outline with developed sections

5. CREATING AND MODIFYING PROPELLER FE MODELS

5.1 FE Grid Proportions

This step is performed after the blade geometry has been defined as described in Chapter 4, or if a PREFX.T16 file from a previous run exists, where PREFX is a character string prefix name used to identify the database files.

Figure 5.1 shows the screen when **FE Grid Proportions** option is selected from the **Create/Modify** submenu.

Figure 5.1: FE Grid Proportions Submenu

The functions of the buttons on this screen are described below:

Exit: Used to exit this submenu.

GenInfo: Used to describe general information about the grid proportion data.

Blade Data: Used to view or edit the radius, pitch, chord, skew and rake data, if necessary. These are the data created during the blade geometry definition stage.

Chord Frac Used to view/edit chord fraction data.
Back Offset: Used to view/edit back offset data, if necessary.
Front Offset: Used to view/edit front offset data, if necessary.
FE Grid: Used to define the FE grid proportions.

Only the last option, **FE Grid**, is required for entering the finite element grid proportions data. The other options are provided for viewing or editing the blade geometry data available on the PREFX.T16. This is necessary when this file is used as the starting point for prescribing the blade geometry. It may not be necessary to access these options for the case where the blade geometry is defined as described in Section 4.

Figure 5.2 shows the Edit Table for creating/editing the **FE Grid Proportions** data.

CREATE AND EDIT FE GRID

PARAMETER	RADIAL DIR	CHORDWISE DIR
NO. OF FE	6	10
GRID PROPORTIONS	BIASED TIP/ROOT	BIASED LE/TE
NO. OF POSITIONS	N/A	1
1ST FRACTION	2.000E-01	1 000E-03
LAST FRACTION	9.950E-01	1 000E+00
GEOM. FACTOR	1.000E+00	1 000E+00
OPTIMIZATION	OPTIMIZE LE/TE	N/A
FRACTION DATA	N/A	N/A

OK Cancel

Figure 5.2: Edit Table for FE Grid Proportions

Data is required to define the finite element grid in the radial and chordwise directions as shown. The required data are described below.

No. of FE Enter the number of finite elements in the radial and chordwise directions

Grid proportion For defining how the grid is proportioned. The desired option can be selected by toggling until the desired option is shown.

For the radial direction, the toggle options are:

USER DEFINED	For grid to be proportioned by the user
BIASED TIP/ROOT	For grid to be automatically biased towards the root and tip
BIASED TO ROOT	For grid to be automatically biased towards the root
BIASED TO TIP	For grid to be automatically biased towards the tip

For the chordwise direction, the toggle options are:

USER DEFINED	For grid to be proportioned by the user
BIASED LE/TE	For grid to be automatically biased towards leading and trailing edges
BIASED TO LE	For grid to be automatically biased to the leading edge
BIASED TO TE	For grid to be automatically biased to the trailing edge

No. of Positions

This option is active only when the USER DEFINED option is selected above. It is used to define the number of chordwise positions at which fraction radii are to be provided. The available options are:

N/A Not applicable, when **USER DEFINED** is not selected

Enter a number from 1 to 3 where for the radial direction

1 = fraction radii provided at leading edge alone
2 = fraction radii provided at leading edge (first) and trailing edge (second)
3 = fraction radii provided at leading edge (first), trailing edge (second) and mid-chord (third)

For the chordwise direction, enter a number from 1 to 2 where:

1 = chordwise positions are the same for all radial positions

2 = chordwise positions are to be specified for each radial position

1^ST Fraction Enter the first fraction radius in the radial direction, and the first chord fraction in the chordwise direction.

Last Fraction Enter the last fraction radius and chord in the radial and chordwise directions, respectively.

Optimization Available for radial direction only.

This is used to define a parameter for optimizing the finite element grid near the blade tip for better shaped elements at the blade tip. The parameter is set by toggling through the options:

NO. OPTIMIZATION	For no optimization
OPTIMIZE LE	For optimizing leading edge only
OPTIMIZE TE	For optimizing trailing edge only
OPTIMIZE LE/TR	For optimizing both leading and trailing edges

Fraction Data Used to define the fraction when the USER DEFINED option is selected.

If automatic grid proportioning option is selected for either direction, the option N/A (for not applicable) is shown in the appropriate data box. In this case the box is inactive.

If the **USER DEFINED** option is selected in either direction, the option **CREATE** is shown in the appropriate data box. Click on **CREATE** to create the function data. Then a table similar to the one shown in Figure 5.3 is displayed for entering/editing the fraction radii for each of the chordwise positions, or for entering/editing the fraction chord. Enter **OK** when finished to return to the **FE Grid Proportion** Edit table.

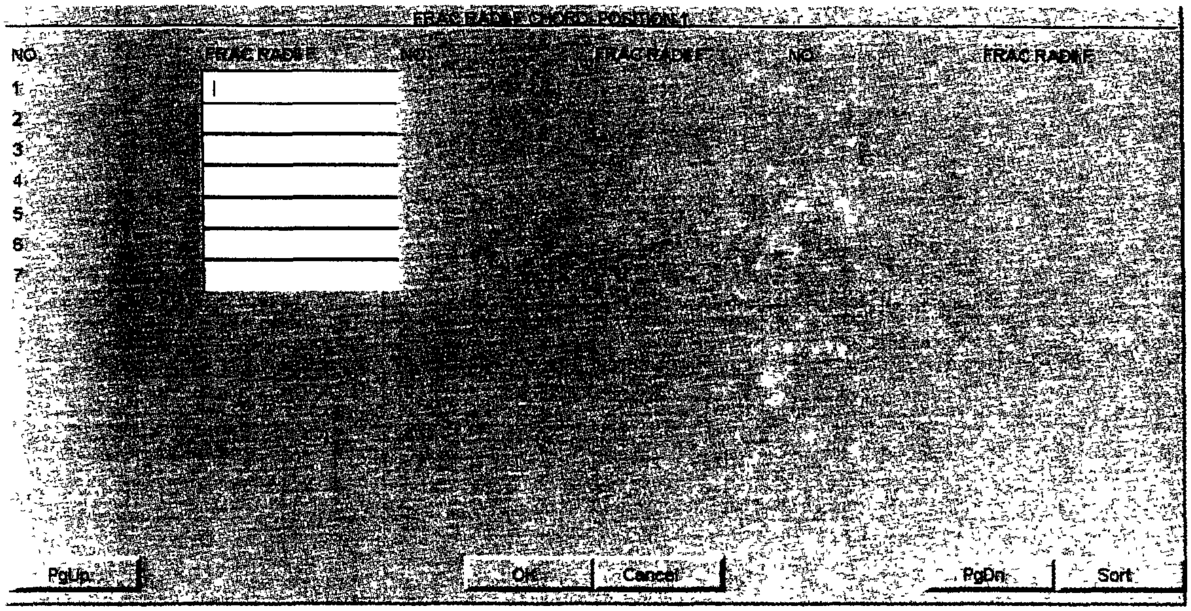


Figure 5.3: Edit Table for Fraction Radii

5.2 Adjusting Peaks of Pressure Loads

This step is to be performed if a static or transient analysis is to be performed. PVAST assumes that hydrodynamic loading on the propeller blade will be available on a file PREFX.LIN, where PREFX is the file prefix name used to identify database files for the current analysis. This file is generated by the hydrodynamic load computation program PUF2 [3]. If this load is not generated by this program, then it must be supplied on the .LIN file. The format of this file is described in Appendix A.

The purpose of this step is to adjust the applied pressure loading to reduce pressure spikes along the leading and trailing edges from which loading errors develop. The smoothed chordwise pressure distributions produce the same thrust and spindle torque as the original pressure distribution.

Figure 5.4 shows the **Adjust Peaks of Pressure Loads** submenu. The user is required to select the form in which the pressures at the face and back of the blade are to be handled. The front and back pressures are either provided as input or as net pressure values.

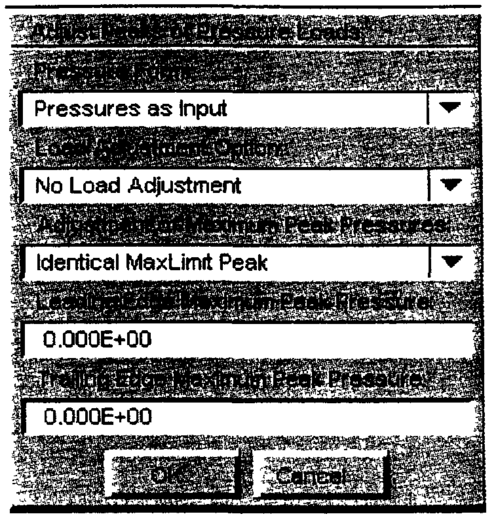


Figure 5.4: Adjust Peaks of Pressure Loads

This menu uses a combination of toggle switches, edit tables for entering or modifying the data. The required data entries are explained below:

Pressure Form: The first data box is used to specify the form of the pressure load. Use the toggle switch to select the option – either the face and back pressures are provided as input, or converted to net pressures.

Load Adjustment Option

This option is used to select the edge(s) where pressure peaks are to be adjusted, using the toggle switches. The options are:

- No Load Adjustment;
- Adjust Leading Edge only; and
- Adjust both the Leading and Trailing Edges.

Select the appropriate option.

Adjustment of Maximum Peak Pressures: This option specifies the maximum peak pressures, using the toggle switch.

The options are:

- Reduce Peaks to Identical Maximum Limits at Both Edges, if required;
- Adjust With no Limits;
- Adjust Using Maximum; and
- Reduce Peaks to Maximum Limits for the Pressure Curves

Select the appropriate option.

Leading Edge Maximum Peak Pressure: This option is for entering the maximum peak pressure at the leading edge, if required. Enter the value of the peak pressure at the leading edge.

Trailing Edge Maximum Peak Pressure: This option is for entering the maximum peak pressure at the trailing edge, if required. Enter the value of the peak pressure at the trailing edge.

Modify Face Pressures: Used to modify the face pressures, when the option to **Reduce Peaks to Max Limits for Curves** is selected above. An edit table will appear as shown in Figure 5.5 for entering the data for each radial position.

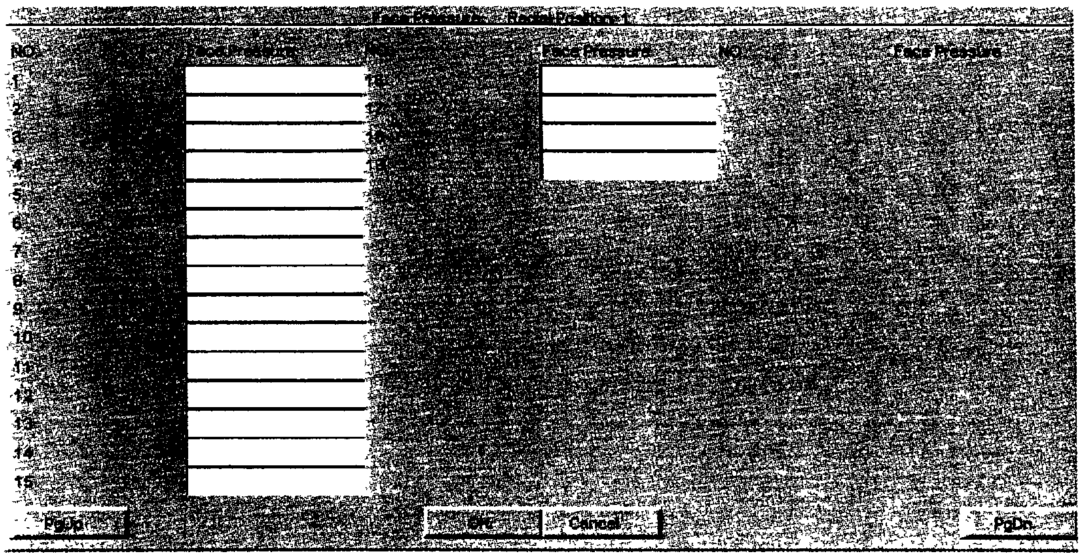


Figure 5.5: Edit Table for Pressure Load Curve Data

Modify Back Pressures: This option is used to modify the back pressures, when the option to Reduce Peaks to Max Limits for curves is selected above. An edit table similar to the one in Figure 5.5 appears for editing the back face pressure curve data.

5.3 Nodal Pressures and Drag Forces

This option is required if a static or transient dynamic analysis is to be performed. It is performed to transform the static or dynamic propeller blade data (available on PREFX.LIN or PREFX.LAD file) into a format that is required for generating the finite element load data after a finite element mesh of the propeller blade has been created. This option is to be performed after the option to **Adjust Pressure Load Data.** However, if no adjustment to the pressure load is required, then the user can perform this option directly. No user input is required. The required data is read from the .LIN or .LAD file.

5.4 Root Model Generation

This step is required for propeller blade models with fillet and palm or hub. The step is performed to create or edit data for defining the root region, consisting of the fillet and palm or hub. The data generated is stored on the PREFX.CDR file under the header "ROOT", for subsequent use by the modules that generate the root models. Figure 5.6 shows the submenu that appears when the option is selected.

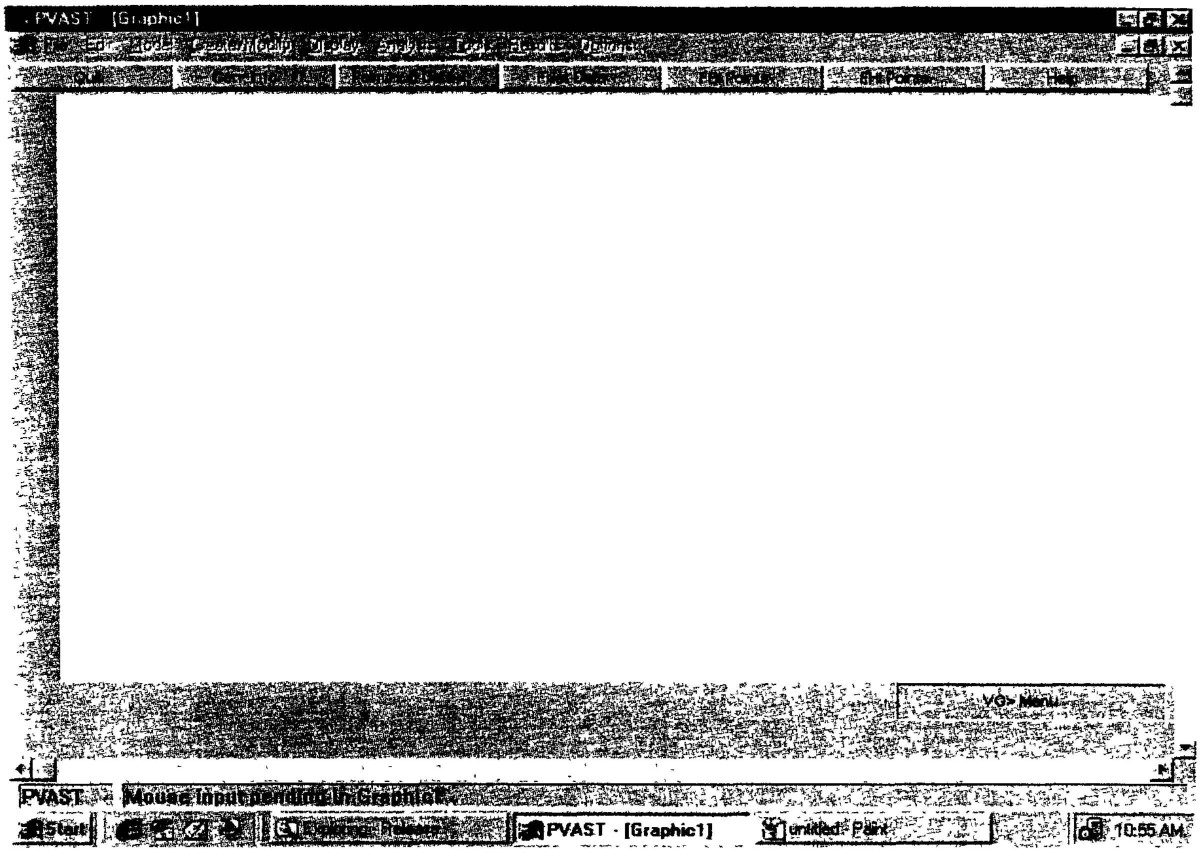


Figure 5.6: Root Model Submenu

The main items on this submenu and their functions are as follows:

Exit:	Used to exit the submenu when finished
Gen Info:	For defining general information on the root model
Palm/Hub Data:	For defining the palm or hub data as the case may be
Fillet Data:	For defining the fillet data
FBI Points:	For defining the fillet-to-blade interface data
FHI Points:	For defining the fillet-to-hub interface data
Help:	For help definitions on the root model

Data under **Gen Info, Palm/Hub Data** and **Fillet Data** are required for all cases in which root models are required. The data under **FBI Points** and **FHI Points** may or may not be required, depending on the options selected in the **Gen Info** section. Details of the input data required for each of the items are provided below.

5.4.1 Exit

Click this button to exit the root generation option, when the data creation is complete. The program will then use the data defined to generate the root model.

5.4.2 Gen Info

The edit table for creating/edit the root general information is shown in Figure 5.7 below:

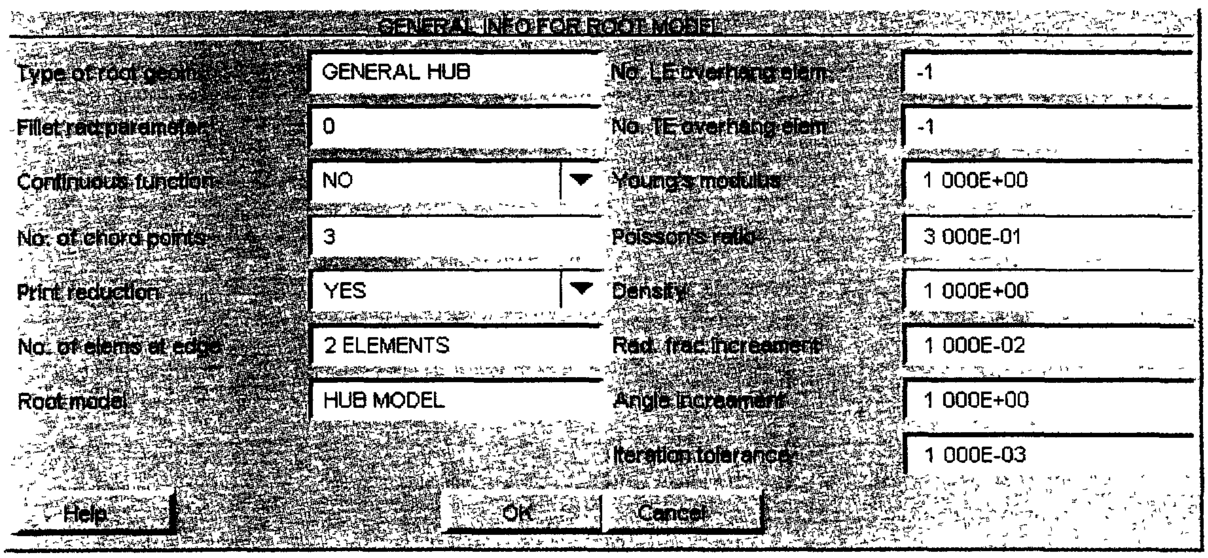


Figure 5.7: Edit Table for General Information on Root Model

User response is required in the two columns beside the data labels. The input data required are summarized below:

Type of Root Geom: Used to define the type of the root model. Click on the box to the right of this label to toggle between **GENERAL HUB** and **PALM GEOM** to chose a hub or palm model.

Fillet Rad Parameter: Used to define a fillet radius parameter

= 0, for single radius fillet (same for face and back), when the radius is constant along chord

= 1, single radius fillet (different for face and back), where the radius varies along chord.

= 2, two radii fillet (same for face and back) based on local blade thickness, as shown in Figure 5.8, where the parameters AK1, AK2 and AK3 are constant along chord.

= 3, two radii fillet (different for face and back), as shown in Figure 5.8, where the parameters AK1, AK2 and AK3 vary along chord.

Enter the number corresponding to the desired option.

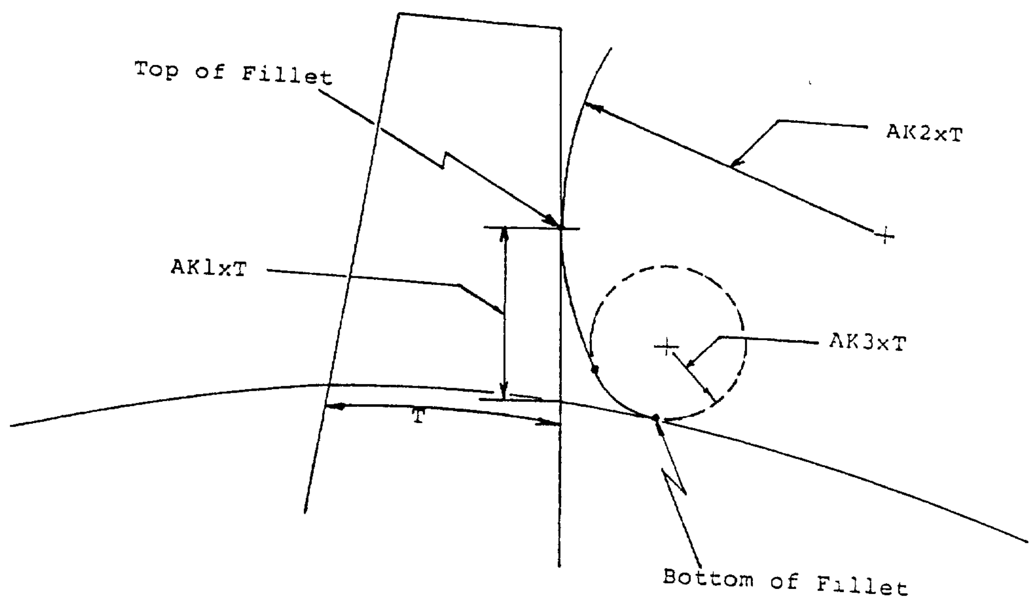


Figure 5.8: Two Radii Fillet at Root

Continuous Function: This parameter is used to set the option to generate a continuous function fillet. Click on the box to the right of this label to toggle between **NO** and **YES**. The option is required only when the fillet radius parameter 2 or 3 is selected above.

No. of Chord Points: For defining the number of chordwise stations at which fillet data is specified. Maximum number of chordwise points is 17. Enter the required number in the box to the right of this label. (Valid when (a) the **Fillet rad parameter** is 1 or 3; (b) the **Fillet rad parameter** is 2 and the continuous function option is on **YES**).

Print Reduction: For reducing the printout of the hub geometry and finite element data in the line printer file. Click the input box to toggle between **YES** and **NO**.

No. of Elems at Edge: For defining number of elements along fillet edge. The options are **2 ELEMENTS** and **3 ELEMENTS** as shown in Figure 5.9(a) and Figure 5.9(b), respectively. Click on the input box and select the appropriate option by toggling through the options.

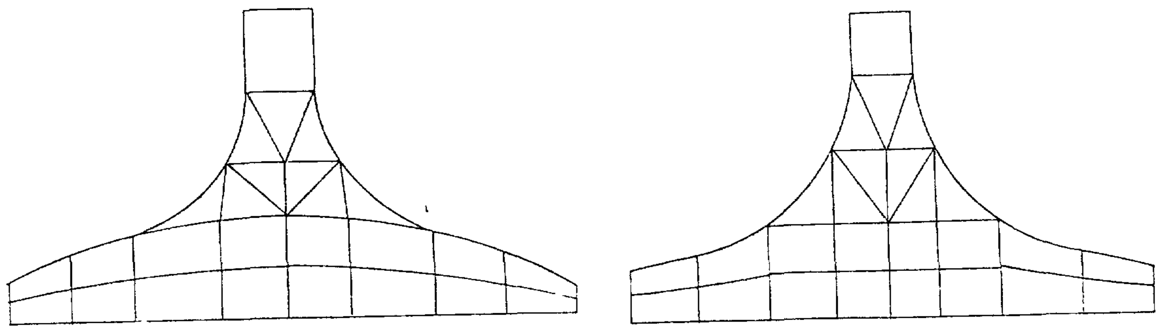


Figure 5.9: Fillet Representations

Root Model: This option is automatically set based on the type of root geometry selected.

No. of LE Overhang Elem: Specify the number of elements in the blade leading edge overhang (usually 1 or 2). The responses are:

- 0, for no overhang.
- A negative number (-1, or –2) if the palm or hub overhangs; or
- A positive number (1 or 2) if the blade overhangs. See Figure 5.10.

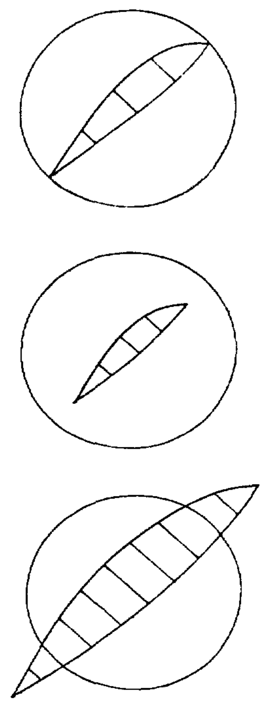


Figure 5.10: Location of Root Base End Relative to Leading and Trailing Edges of the Blade

No. of TE Overhang Elem: Specify the number of elements in the blade trailing edge overhang, as described above.

Young's Modulus: Enter the Young's modulus (force/length^2) of the root material.

Poisson's Ratio: Enter the Poisson's ratio of the root material.

Density: Enter the density (force-time^2/length^4) of the root material.

Rad Frac Increment: Enter fraction of full blade radius used in iteration procedure for locating the fillet. Defaults to 0.01.

Angle Increment: Enter the angle increment (in degrees) used in integration procedure for locating fillet. Defaults to 1.0.

Iteration Tolerance: Enter the tolerance to which iteration is carried out. Defaults to 0.001.

5.4.3 Palm/Hub Data

The input data required depends on the root model selected (palm or hub model). For palm geometry the following screen appears:

5.4.3.1 Palm Model

For palm geometry, the edit table shown in Figure 5.11 appears.

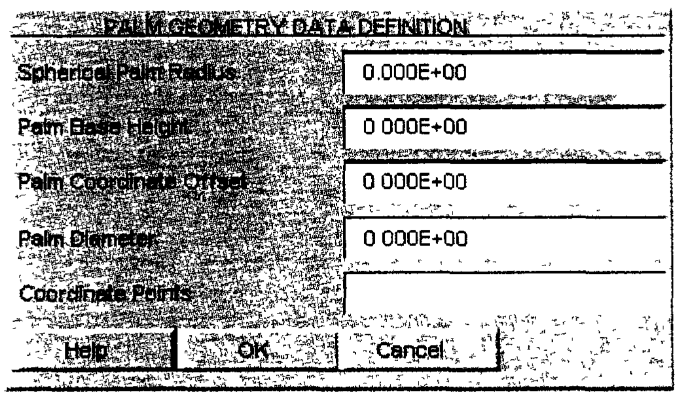


Figure 5.11: Edit Table for Palm Model

The parameters are as described below.

Spherical Palm Radius: If the palm is known to be spherical, enter the spherical radius in length units. Otherwise, enter 0 to define palm geometry by co-ordinate points on a 16-noded patch.

Palm Base Height: Enter the palm base height (length).

Palm Co-ordinate Offset: Enter the palm co-ordinate offset for spherical palm model only. See Figure 5.12.

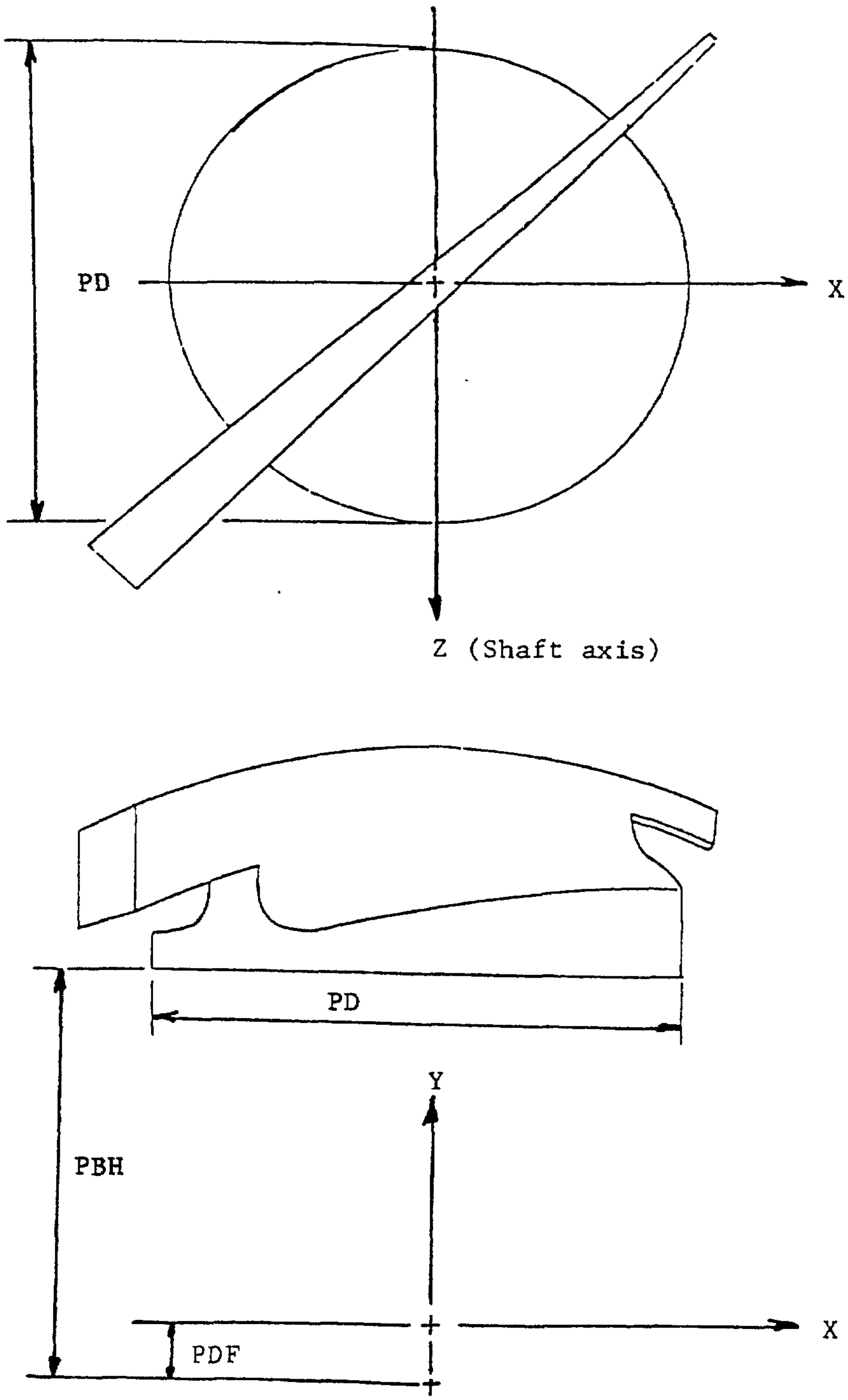
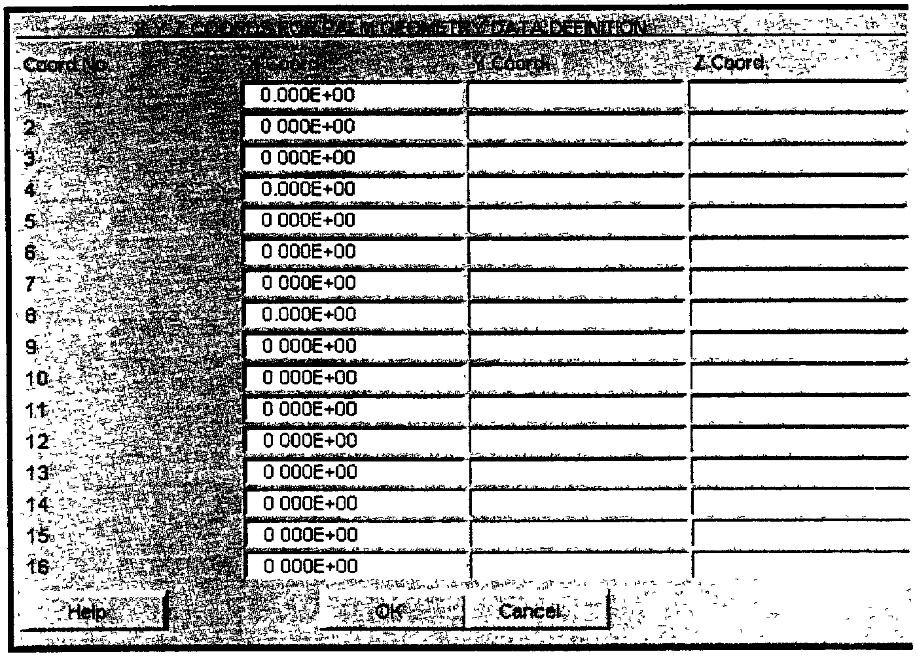


Figure 5.12: Illustration of Palm Coordinate Offset for Spherical Palm

Palm Diameter: Enter the palm diameter (length).

Co-ordinate Points: Valid only if spherical palm radius of 0 is entered (general palm definition). Click on **CREATE** to create the co-ordinates of the data points. An edit table appears for entering the patch for describing the palm geometry. The edit table is shown in Figure 5.13 below. Click **OK** when finished.



(a) Edit Table

Figure 5.13 : Defining Palm Surface Geometry (a) Edit Table; (b) Location of 16 Points Defining the Patch

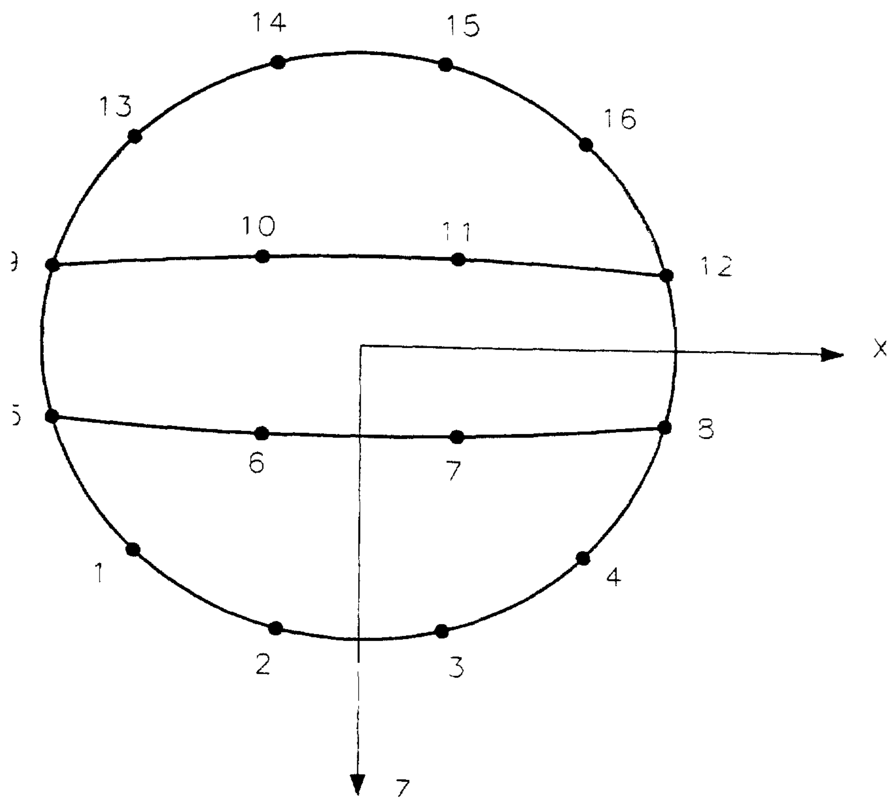


Figure 5.13(b) Location of 16 Points Defining the Patch

5.4.3.2 Hub Model

For hub geometry, the following screen (Figure 5.14) appears:

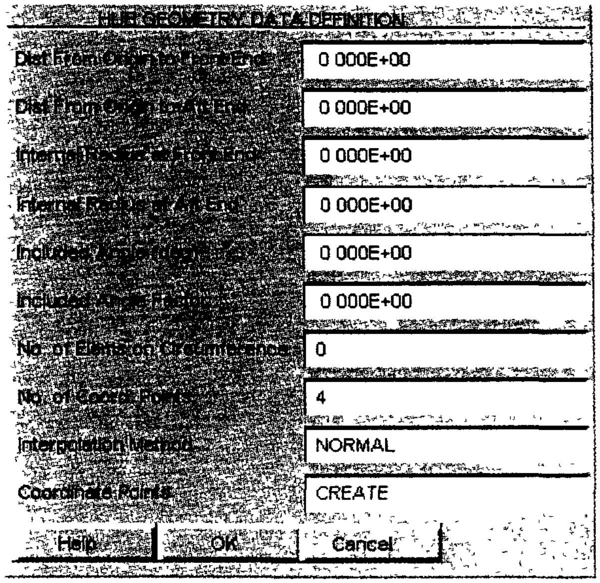


Figure 5.14: Edit Table for Hub Geometry Definition

The required input data are described below.

Dist From Origin To Front End: Enter the distance (length) from the origin to the front end (as positive value).

Dist From Origin to Aft End: Enter the distance (length) from the origin to the aft end (as positive value).

Internal Radius at Front End: Enter the internal hub radius at front end (length). (RID in Figure 5.15 at front end.)

Internal Radius at Aft End: Enter the internal hub radius at the aft end (length). (RID in Figure 5.15 at aft end.)

Included Angle: Enter the included angle for bladed and bladeless hub segment pairs (in degrees). (PHI2 in Figure 5.15.)

Included Angle Factor: Enter the included angle factor for defining the size of bladeless hub segment. Set to 0.0 if no bladeless hub segment is to be generated. (PHIF in Figure 5.15.)

No. of Elems on Circumference: Number of elements in circumferential direction for the bladeless hub segment. Set to 0 if there is no bladeless hub segment. Illustrations of the parameters are provided in Figure 5.15.

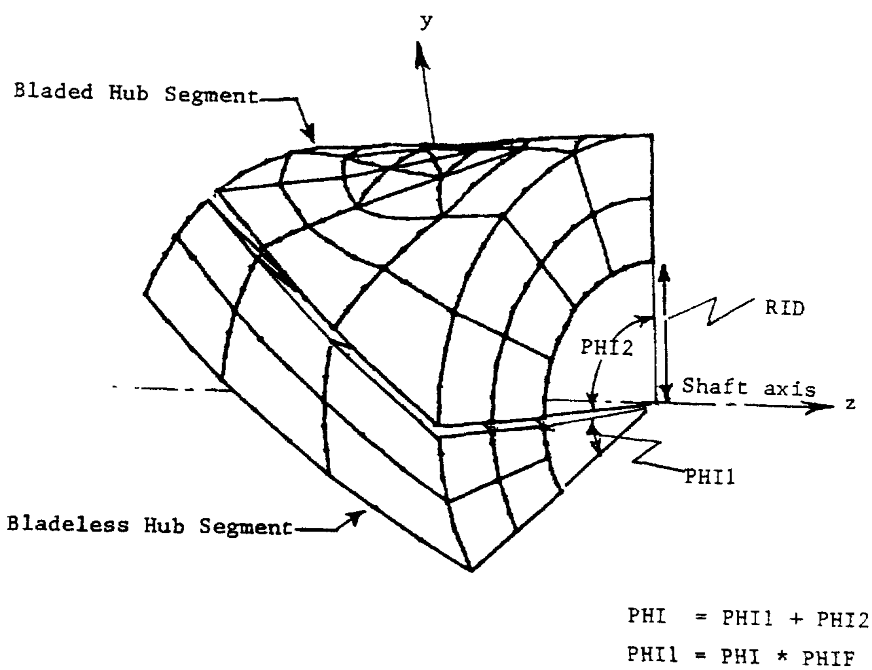


Figure 5.15: Geometric Parameters for Hub Model

No. of Coord Points: Enter the number of points for defining the hub shape. The number should preferably be greater than 4 since a cubic spline is used.

Interpolation Method: Click the input box to select the interpolation method. The options are **NORMAL** and **INTRINSIC** methods of interpolation.

Co-ordinate Points: Click on **CREATE** in the input box to create the Z and Y co-ordinates of the points defining the hub geometry. An edit table (shown in Figure 5.16) appears for entering the data.

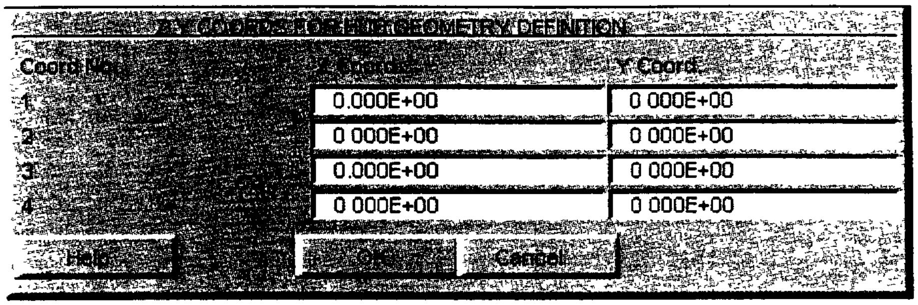


Figure 5.16: Edit Table for Defining Coordinates of Hub Geometry

Enter the Z- and Y- co-ordinates of the points defining the hub geometry. Click OK when finished.
Note that the Z-coordinate is along the shaft axis, and the Y-coordinate along the blade vertical axis as shown in Figure 4.3.

5.4.4 Fillet Data

The required data depends on the fillet radius option selected in the **Gen Info** section.

(a) Single radius fillet (same for face and back), for single fillet (same for face and back), the following edit table (Figure 5.17) appears:

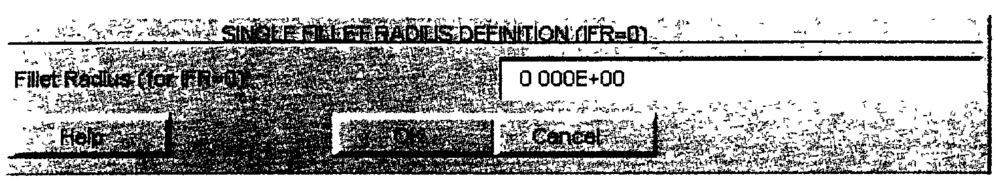


Figure 5.17: Edit Table for Single Fillet Radius Definition

Enter the fillet radius. Click **OK** when finished.

(b) Single Radius Fillet (Different for Face and Back)

For single radius fillet (different for face and back) where radius varies along chord, the following edit table (Figure 5.18) appears for entering/editing the fillet data.

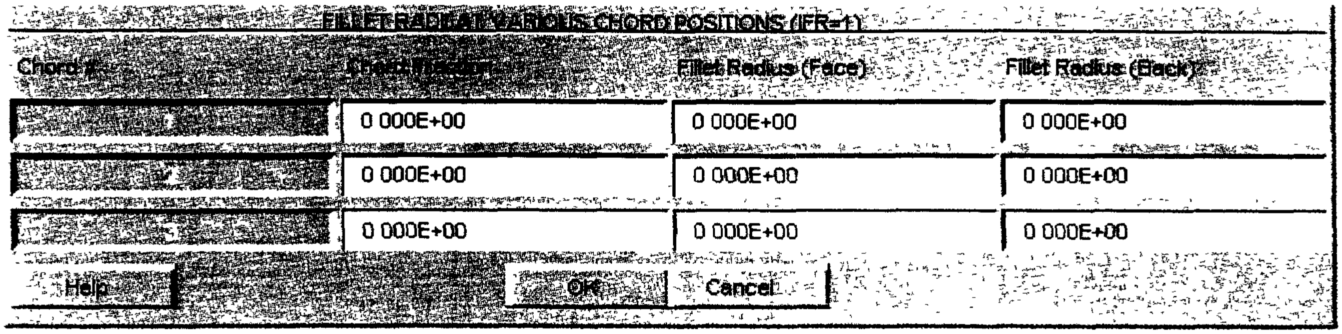


Figure 5.18: Edit Table for Fillet Radii at Various Chord Positions

Enter the chord fractions and the fillet radii at the face and back for the NCST points, where NCST is the number of chordwise points at which the fillet radii are defined, as specified in the **Gen Info** section. Click **OK** when finished.

(c) Two Radii Fillet (same for face and back)

For this option, the following edit table (Figure 5.19) appears for defining the constants for fillet generation.

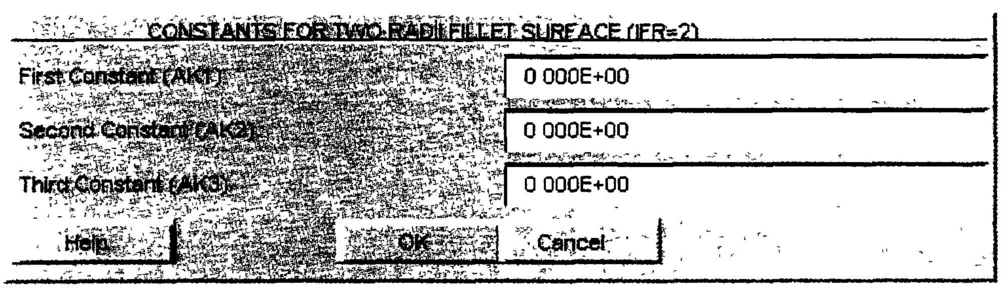


Figure 5.19: Edit Table for Defining Constants for Two Radii Fillet Surface

Enter the constants AK1, AK2, AK3 (see Figure 5.8). Click **OK** when finished.

(d) Two Radii Fillet (different for face and back)

The edit table for this option is shown in Figure 5.20 below:

CONSTANTS FOR 2-RADII FILLET SURFACE (IFR=3)

Chord #	Chord Frac	AK1 (Face)	AK2 (Face)	AK3 (Face)	AK1 (Back)	AK2 (Back)	AK3 (Back)
1							
2							
3							

Help OK Cancel

Figure 5.20: Edit Table for Defining Front and Back Constants for 2-Radii Fillet Surface

Enter the chord fractions and the constants AK1, AK2 and AK3, for the NCST chordwise points for defining the fillet. Click **OK** when finished.

5.4.5 FBI Points

This option is valid only when the continuous function option is selected in the **Gen Info** section. The option is used to define the fillet-to-blade intersection (tangency). Lengths of tangent vectors are defined at the NCST chordwise points. Zero values will result in those tangent lengths being calculated by a best curve fitting algorithm. Figure 5.21 shows the edit table for this option.

LENGTH OF FILLET-BLADE INTERFACE TANGENT VECTORS

Chord #	Chord Fraction	Vector Length(Face)	Vector Length(Back)
1	0.000E+00		
2	0.000E+00		
3	0.000E+00		

Help OK Cancel

Figure 5.21: Edit Table for Defining Tangent Vectors of Fillet-to-Blade Interface

Enter the vector length at the face and back on the NCST points used to define the fillet. Click **OK** when finished.

5.4.6 FHI Points

This option is valid only when the continuous function option is selected in the **Gen Info** section. The option is used to define the fillet-to-hub intersection (tangency). Lengths of tangent vectors are defined at the NCST chordwise positions. Zero values will result in those tangent lengths being calculated by a best curve fitting algorithm. Figure 5.22 shows the edit table for this option.

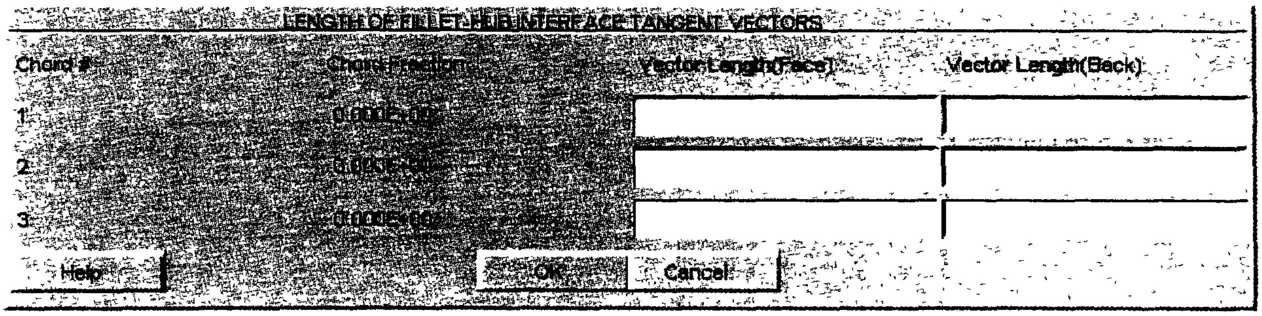


Figure 5.22: Edit Table for Defining Tangent Vectors for Fillet-Hub Interface

Enter the vector lengths at the face and back of the NCST points used to define the fillet. Click **OK** when finished.

5.4.7 Help

The help option provides information on the root data options. The submenu is as shown in Figure 5.23.

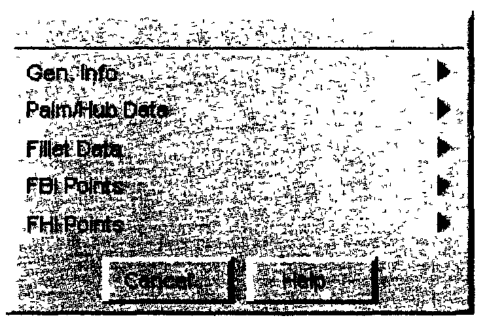


Figure 5.23: Help Submenu for Root Model

Click on the desired option to view the description of that option. Click cancel to exit the **Help** option.

5.5 Adding Bolts and Gaps to Root FE Model

This section is intentionally left blank.

5.6 Blade Finite Element Generation

This option is required to generate the blade finite element model. The option to generate **FE Grid Proportions** must be implemented prior to this step. Also, for models requiring root models, it is necessary to generate the **Root Model** before this step.

The edit table for this option is shown in Figure 5.24 below:

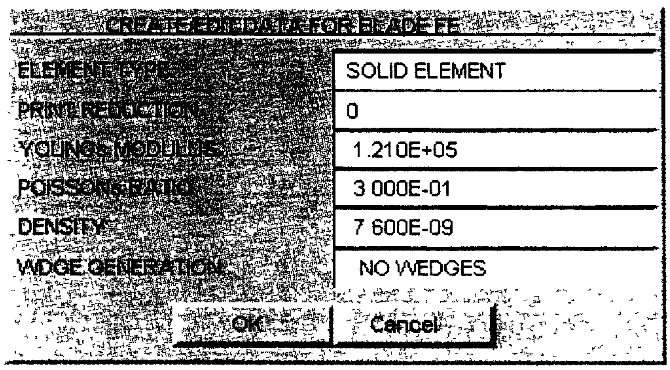


Figure 5.24: Blade FE Edit Table

The following are the required data entries for this option.

Element Type: For defining the type of finite elements to use. Click the input box to toggle between **SOLID ELEMENT** and **SHELL ELEMENT**.

Note:
When shell elements are used, the thickness at the edges should not be zero at the end points. If this is the case, the grid proportions should be set so that start and end a small distance from the leading and trailing edges of the blade. Otherwise solid elements with wedges should be used.

Print Reduction: Enter 0 for no reduction of printout in line printer file.
Enter 1 if printout reduction is required.

Young's Modulus: Enter the Young's modulus of the blade material (force/length^2).

Poisson's Ratio: Enter the Poisson's ratio of the blade material.

Density: Enter the density of the blade material (force-time^2/length^4).

WDGE Generation: This data is required only when solid elements are used to model the blade. The wedge generation parameter is used to trigger the generation of wedge elements by collapsing the solid elements at the leading and trailing edges of the blade. Click the corresponding input box and toggle between **NO WEDGES** and **FORM WEDGES** to select as required.

Click **OK** when finished. The program will now use the data generated to create the blade finite element model.

5.7 Substructured Blade-Root FE Model

This step is required to generate the substructured blade-root finite element models, for blade-fillet-palm, blade-fillet-hub or multi-blade-hub models. **Blade FE** and **Root FE** models must be generated prior to this step. The following submenu (Figure 5.25) appears for defining or editing the data required to generate the substructured FE model.

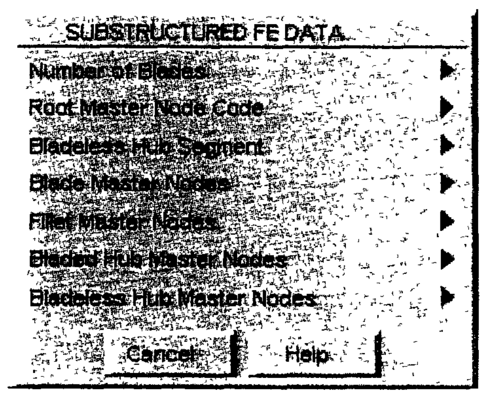


Figure 5.25: Substructured FE Data Submenu

Descriptions of the submenu items and required user responses are provided below.

Number of Blades: For defining the number of blades in the model. An input box appears when this item is selected. Enter the required number of blades (from 1 to 8). Enter **OK** when finished.

Root Master Node Code: This item is used to specify the manner in which master nodes are determined. A pick menu with the choices *Interface Nodes Only*, and *Interface and Side Nodes* is presented. Click on the required item to select the root master node code. The selected item will be highlighted. Click **OK** when finished.

Note:
If the *Interface Nodes Only* option is selected, only master nodes at interfaces between the blade and fillet as well as between fillet and palm/hub are determined. If the *Interface and Side Nodes* option is selected, master nodes at interfaces between blade and fillet, at the interface between the fillet and palm/hub; and along sides of the hub are determined. This is necessary when modelling the complete hub.

Bladeless Hub Segment: This item is for specifying the generation of a bladeless hub segment. A **YES/NO** toggle switch is presented for selecting the appropriate option. Click **OK** when finished.

Blade Master Nodes: This option is used to specify user defined master nodes, which will be used in addition to those automatically generated by the program for the blade. A pick menu appears with the options:

- No Additional Nodes
- Create Additional Nodes
- Use Previous Additional Nodes

Click on the appropriate item to select the desired option.
Select *No Additional Nodes* if no additional master nodes are required.
Select *Create Additional Nodes* to create additional master nodes. This option is not functional in Version 7.3.
Select *Use Previous Additional Nodes* if user specified master nodes for a previous blade are to be used. This option is also not functional in Version 7.3.
Repeat the data entry for all blades.

Fillet Master Nodes: This option is used to specify user defined master nodes, which will be used in addition to those automatically generated by the program, for the fillet. A pick menu similar to the one described under **Blade Master Nodes** is presented and the data entry is also similar.

Bladed Hub Master Nodes: This option is used to specify user defined master nodes which will be used in addition to those automatically generated by the program, for the bladed hub segment. A pick menu similar to the one described under **Blade Master Nodes** is presented and the data entry is also similar.

Bladeless Hub Master Nodes: This option is required only if the bladeless hub segment generation option was selected. The option is used to specify user defined master nodes, which will be used in addition to those automatically generated by the program, for the bladeless hub segment. A pick menu similar to the one described under **Blade Master Nodes** is presented and the data entry is also similar. Click **OK** when finished.

5.8 Loads on FE Model

This step is required to generate the finite element load on the blade FE mesh. No additional input data is required. The program automatically uses the load data on the PREFX.LIN or .LAD file, and the FE model of the blade. It is therefore essential that the steps **Nodal Pressures and Drag Forces** and **Blade FE Model** be executed prior to this step. Also, if centrifugal stiffening is required, the rotational velocity (cycles/sec) should first be defined. This is done during the creation/modification of **Properties**, as described in Section 5.15.

The program generates the finite element load data and prints the message "Finished Generating FE Loads".

5.9 Boundary Conditions

This step is required to generate the boundary conditions (BC) on the FE model. Finite element models of the blade and root (if present) should be generated prior to the execution of this step. Since the boundary conditions generation is accomplished via interactive graphics, it is also necessary to display the finite element model prior to the generation of boundary conditions. Along with the finite element models, the program automatically generates default boundary conditions which can be used as the starting point for BC generation. For instance, for blade FE models, the program applies fixed boundary conditions to all nodes at the base of the blade. These BCs can be modified or deleted, or additional BCs can be created, as required. The editing or creation of BCs is accomplished by cursor selection of single or multiple nodes on the model, using a mouse. A detailed description of the procedure is available in the MGDSA Manual [8].

5.10 Gap Elements

This section is intentionally left blank.

5.11 Fluid FE Models

This step is required to generate fluid finite element models in cases where the analysis of the blade in water is required. A **Blade FE** model of the blade must be generated prior to this step. The following submenu (Figure 5.26 (a)) appears for creating or editing the data required to generate the fluid FE model.

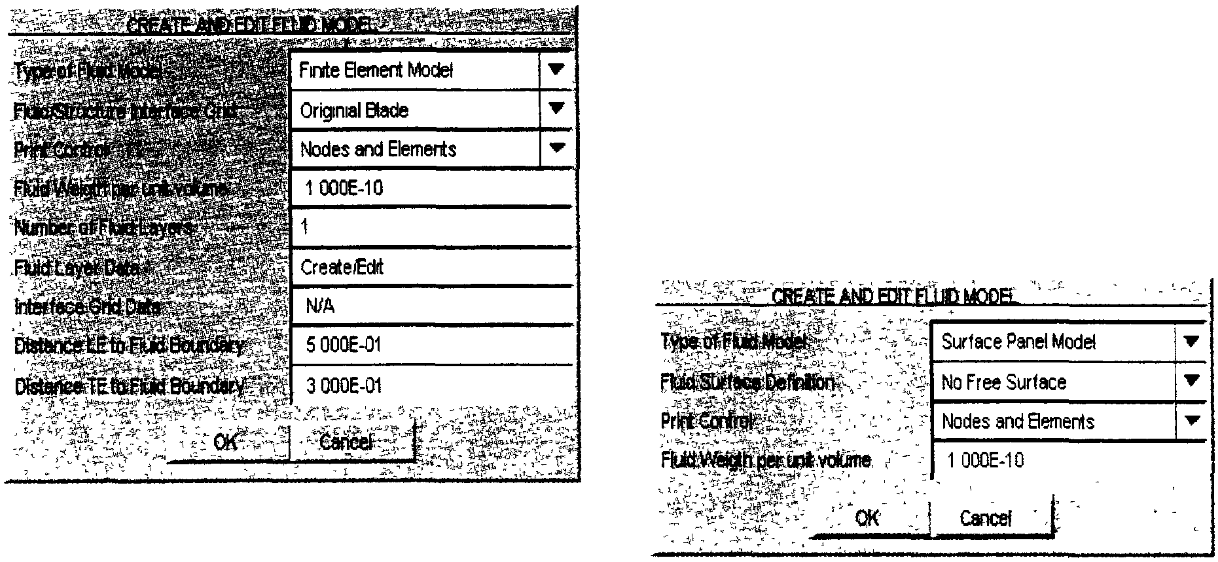


Figure 5.26: Fluid FE Model Submenu (a) 3-D Finite Element Model, (b) Surface Panel Model

Descriptions of the submenu items and the required user responses are provided below.

Case 1: Full 3-D Finite Element Model

Type of Fluid Model: Used to specify the type of fluid finite element model. There are two types of fluid models (a) a full 3-D finite element model and (b) a surface panel. Click on the toggle to the required option. If the type of fluid model is changed from full 3-D finite element model to surface panel model, the user will be required to confirm the choice, because the data for surface panel method is different and a new submenu (as shown in Figure 5.26) will be shown. In this case data already entered under the finite element model may be lost.

Fluid/Structure Interface Grid: This option is used to specify the grid to be used for the fluid/structure interface. The options are **Original Blade** and **User Defined**. The **Original Blade** option uses the existing blade finite element grid. The

User Defined option enables the user to specify any other desired grid. This is useful for reducing the finite element mesh as shown in Figure 5.27. Use the toggle switch to select the desired option.

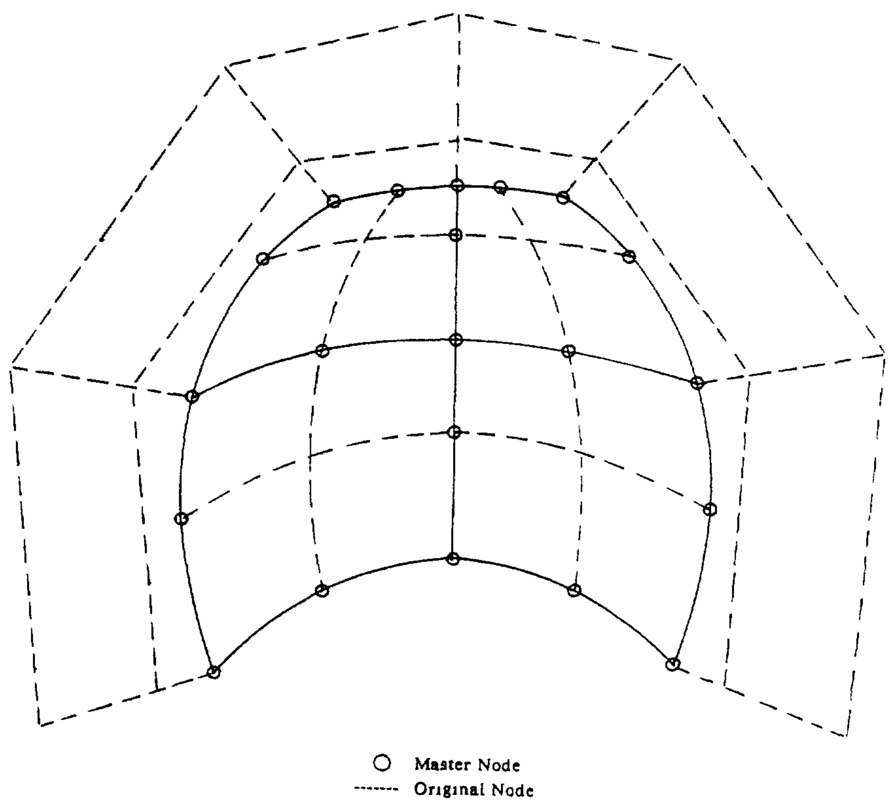


Figure 5.27: Reduced FE Mesh for a Typical Propeller Fluid Model

Print Control: This parameter is used to control the printout of the fluid finite element model data in the line printer file. If **Nodes** and **Elements** is selected, the nodal co-ordinates and element nodes will be printed. If **Reduced Printout** is selected then these will not be printed. Use the toggle switch to select the required option.

Fluid Weight per Unit Volume: Enter the fluid density in weight per unit volume/g (force-time^2/length^4).

Note:
The appropriate values of densities for seawater and fresh water are in metric and imperial units:

seawater: 0.103E-8 (metric) ; 0.989E-4 (imperial)
fresh water: 0.102E-8 (metric) ; 0.975E-4 (imperial)
The units of density in metric and imperial units are Ns^2/m^4 and lb-s^2/in^4, respectively.

Number of Fluid Layers: Enter the number of fluid layers (NLAYER) on the propeller face and back (the total number of layers is 2xNLAYER. Figure 5.28 shows the case with NLAYER=2.

Fluid Layer Data: Used to enter the depth of the fluid layers. Click on **Create/Edit** and an edit table appears to enter the depths of the fluid layers, expressed in a fraction of the full blade radius. Enter the layer data and click **OK** when finished. See Figure 5.28.

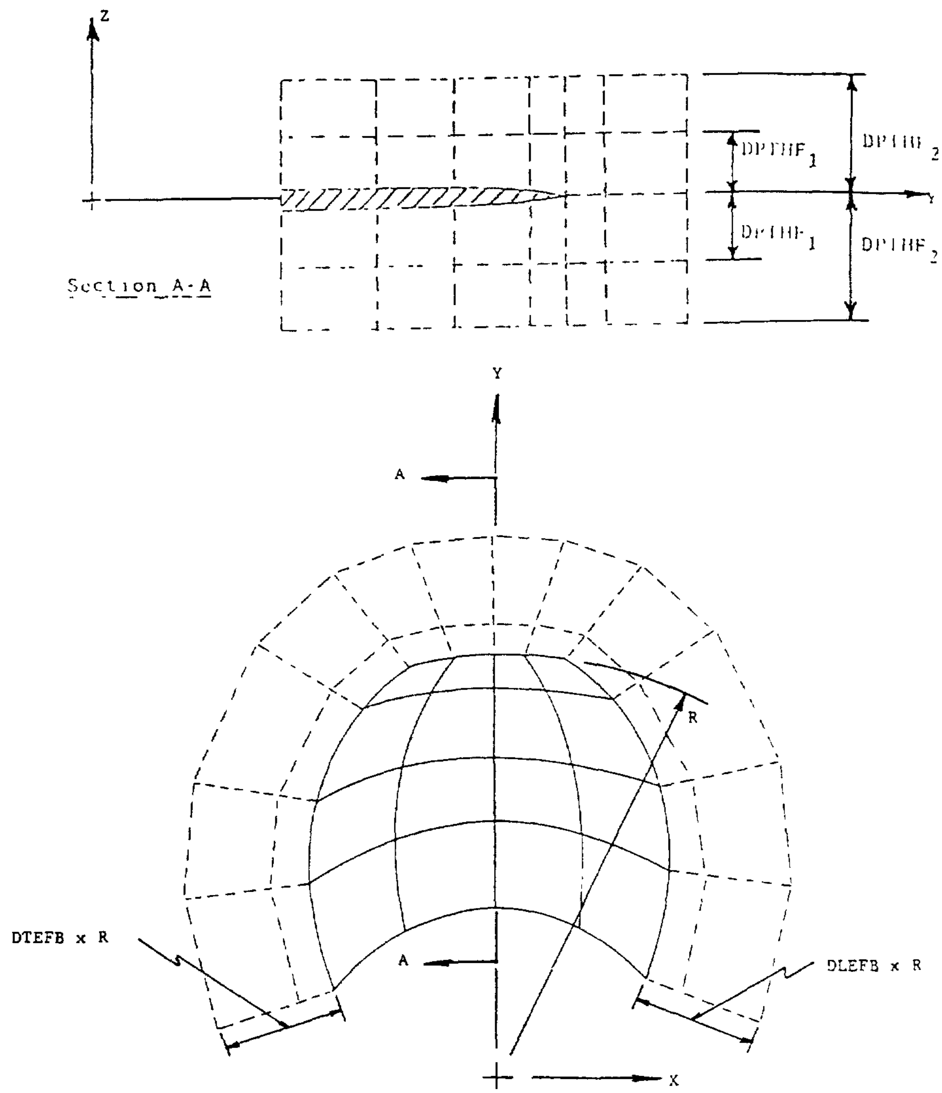


Figure 5.28: Fluid Geometry Definitions

Interface Grid Data: This option is used for defining the user defined interface data. It is only active when the **Fluid/Structure Interface Grid** option selected above is the **User Defined** option.

Click on **Create/Edit** to create the interface data. Then, the following submenu (Figure 5.29) appears for defining the interface grid data.

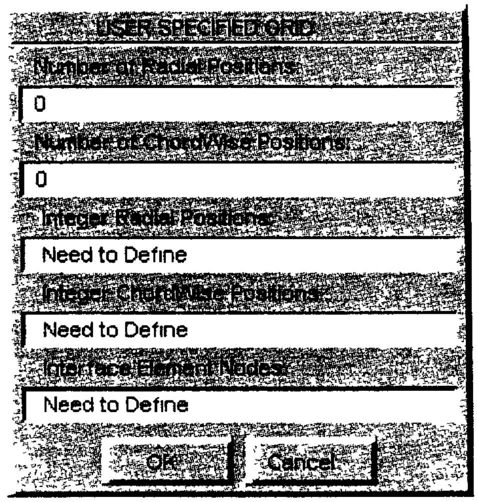


Figure 5.29: User Specified Grid

The required entries are:

- The integer radial and chordwise array locations, selected from the radial and chordwise positions, respectively, used to define the original blade elements; and
- The fluid/structure interface element numbers, defined as shown in Figure 5.30.

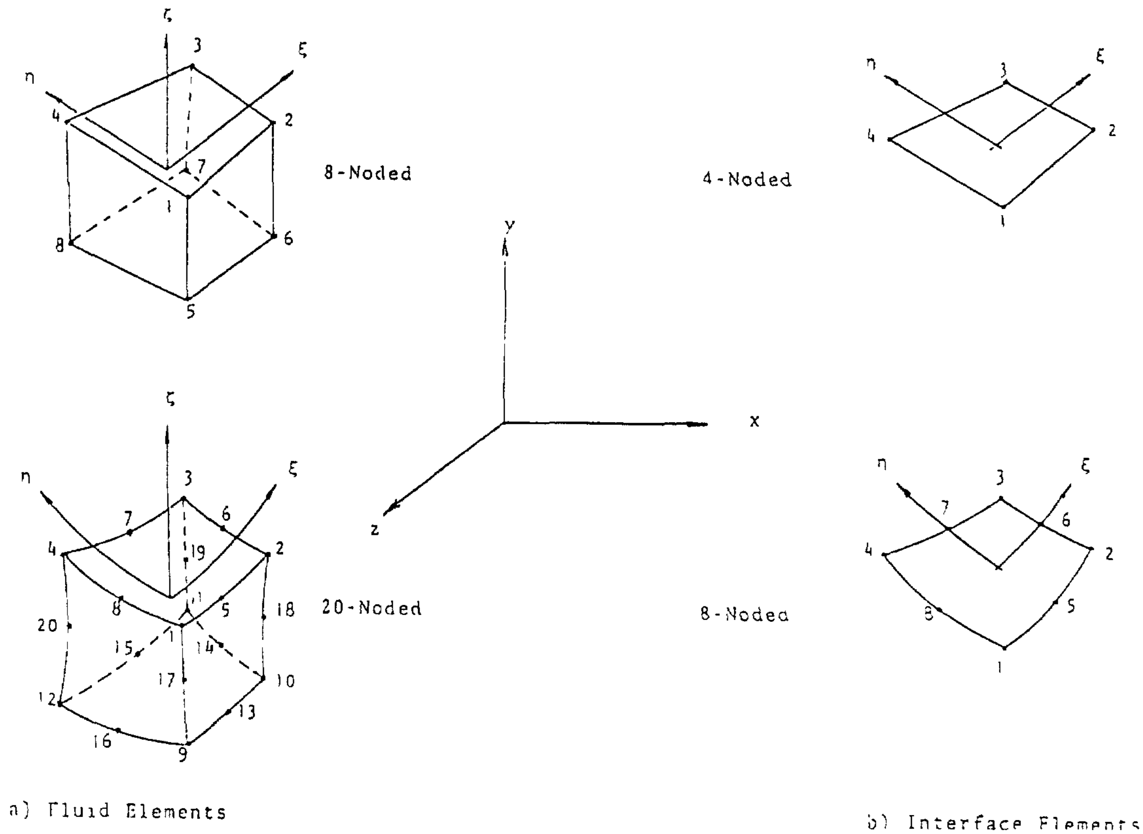


Figure 5.30: Fluid Finite Elements

Distance LE to Fluid Boundary: Enter the distance from the leading edge to the fluid boundary expressed as a fraction of the full blade radius (see Figure 5.28).

Distance TE to Fluid Boundary: Enter the distance from the trailing edge to the fluid boundary expressed as a fraction of the full blade radius (see Figure 5.28).

Case 2: Surface Panel Model

Type of Fluid Model: This option can be used to specify the type of fluid finite element model. The option is selected by clicking on the toggle switch. If the type of fluid model is changed from surface panel to full 3-D finite element model, the user will be required to confirm the choice as this selection invokes the finite element model submenu which is different from the surface panel submenu.

Fluid Surface Definition: Used to specify the fluid surface condition. The options are **No Free Surface** and **X-Y Free Surface**. Click the toggle switch to select the required option. If the X-Y Free Surface Option is selected, then free surface effects in the X-Y plane are accounted for.

Fluid Weight per Unit Volume: This is as described in Case 1.

5.12 Generating Input File for VAST FE Analysis

This step is required to generate the input file for finite element analysis via the VAST FE solver. The step should be performed after all the various other steps have been implemented. The data created is stored in a file called PREFX.USE which is required by VAST. Figure 5.31 shows the submenu for entering the required data. Descriptions of the data and user responses are provided below.

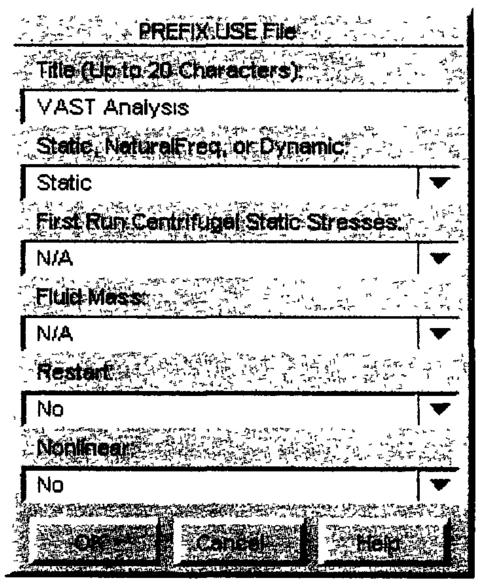


Figure 5.31: VAST Input File Creation Submenu

Title Enter a title to identify the analysis. Up to 20 characters are allowed.

Analysis Type: Used to specify the type of analysis. The options are **Static, Natural Frequency** and **Dynamic** analysis. Click on the toggle switch to select the desired option.

First Run: Centrifugal Static Stresses: This option is active when centrifugal stiffening option is selected. Recall that the centrifugal stiffening option is selected through the **PROPERTIES** option of the **Create/Modify** submenu. The option is used to specify if this is the first run for computing the stresses due to centrifugal stiffening. If centrifugal stiffening is inactive then the input box is filled with N/A for not available.

Fluid Mass: This option is required only when an analysis of the blade in water is to be performed. Click the input box and toggle to **YES** to activate this option. It is assumed that a fluid model of the blade has been developed prior to this step.

Restart: This option is used to specify if the current analysis is a restart. Click the input box to toggle to **YES** or **NO** as required.

Nonlinear: This option is used to specify if a nonlinear analysis is to be performed. Click the input box to toggle to **YES** or **NO** as required.

The program is designed to automatically generate most of the input data using default values for the options selected above. However, user response is still required when reasonable default values cannot be used. The subsequent responses depend on the options selected above. These are described below for **Static, Natural Frequency** and **Dynamic** analyses.

Static Analysis

Figure 5.32 shows the submenu for entering additional static analysis data.

Figure 5.32: Static Analysis Submenu

The submenu is used to select the type of load for the analysis. The options are described below.

Element/Concentrated Loads Loads (already defined in the load step) are due to element and/or concentrated loads.

Support Motion This option is required if loads are due to support motion. In this case the following submenu (Figure 5.33) appears for defining the support motion data. Detailed descriptions of the parameters are provided in the VAST Version 7.3 Manual [7].

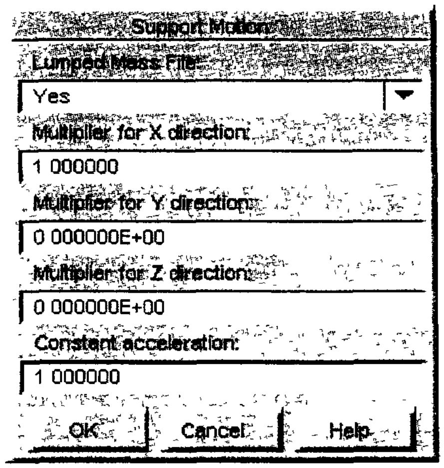


Figure 5.33: Support Motion Submenu

Natural Frequency Analysis

Figure 5.34 shows the Natural Frequencies submenu.

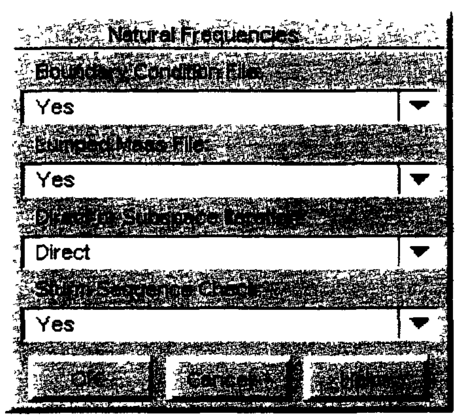


Figure 5.34: Natural Frequency Analysis Submenu

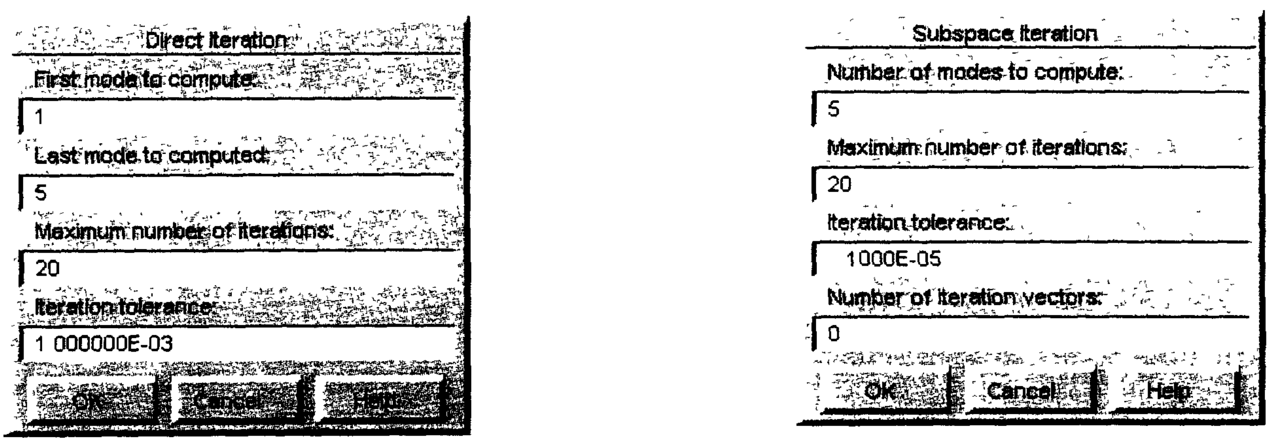
The required responses are:

Boundary Condition File: Select **YES** or **NO**, depending on whether boundary conditions are available on the boundary conditions file (PREFX.SMD) or not.

Lumped Mass File: Select **YES** or **NO**, depending on whether mass modification data is available on the mass modifications file (PREFX.MMD) or not.

Direct or Subspace Iteration: Used to select the method for computing eigenvalues. The options are **Direct** or **Subspace** iteration methods. Select the required option by clicking the toggle switch.

Sturm Sequence Check: Select **YES** or **NO** to activate/deactivate sturm sequence check. Click **OK** when finished. Depending on the iterative computational method selected under **Direct** or **Subspace** Iteration, either of the following submenus (Figure 5.35) will appear for entering the number of modes and iteration tolerances. See Reference [8] for further details.



(a) (b)

Figure 5.35: Iteration Method Submenus, (a) Direct, (b) Subspace Method

If the **Sturm Sequence Checking** option was selected, then the following submenu (Figure 5.36) appears for entering the eigenvalue shift factor.

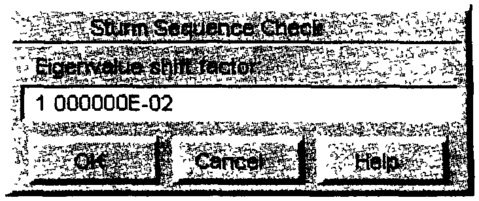


Figure 5.36: Sturm Sequence Check Submenu

Dynamic Analysis

Figure 5.37 shows the dynamic analysis submenu for entering the dynamic analysis data. Detailed descriptions of the data are available in Reference [7].

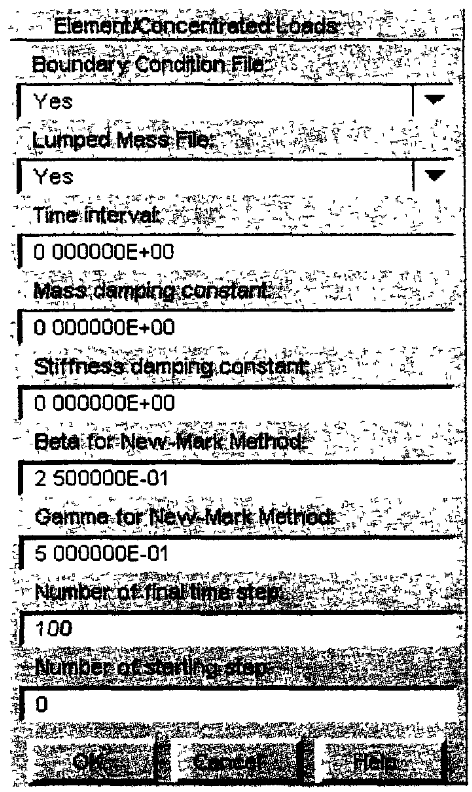


Figure 5.37: Dynamic Analysis Submenu

5.13 Keyway Models

5.13.1 Keyway Geometry

This option is used to generate two-dimensional finite element models for hubs with keyways. Figure 5.38 shows the edit table for defining the keyway geometric data.

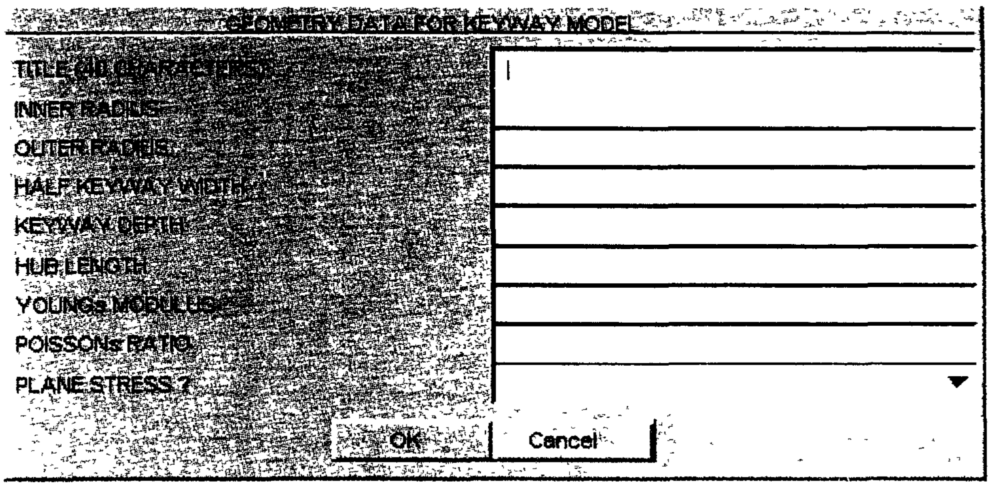


Figure 5.38: Keyway Geometry Data Submenu

The required data entries are described below (see Figure 5.39).

Title: Used to define a title for the model. Up to 40 characters are allowed.

Inner Radius: Enter the inner radius of the keyway (RI) (length).

Outer Radius: Enter the outer radius of the keyway (RO) (length).

Half Keyway Width: Enter one-half keyway width (A) (length).

Keyway Depth: Enter keyway depth (B) (length).

Hub Length: Enter the hub length.

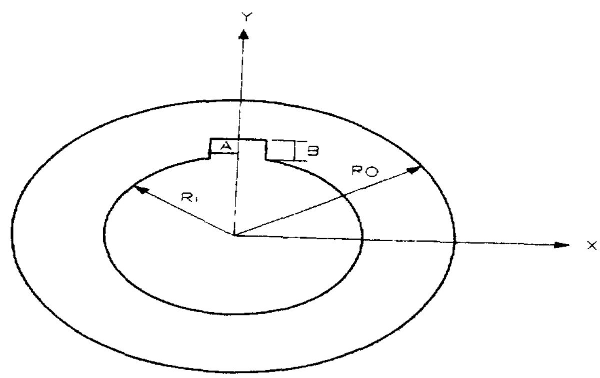


Figure 5.39: Geometry of Hub With Keyway

Young's Modulus: Enter Young's modulus of hub material (force/length^2).

Poisson's Ratio: Enter Poisson's ratio of hub material.

Plane Stress: Toggle to **YES** if plane stress analysis is to be performed and **NO** if a plane strain model is used.

Click **OK** when finished.

5.13.2 Keyway Load Data

Figure 5.40 shows the edit table for defining the keyway load data.

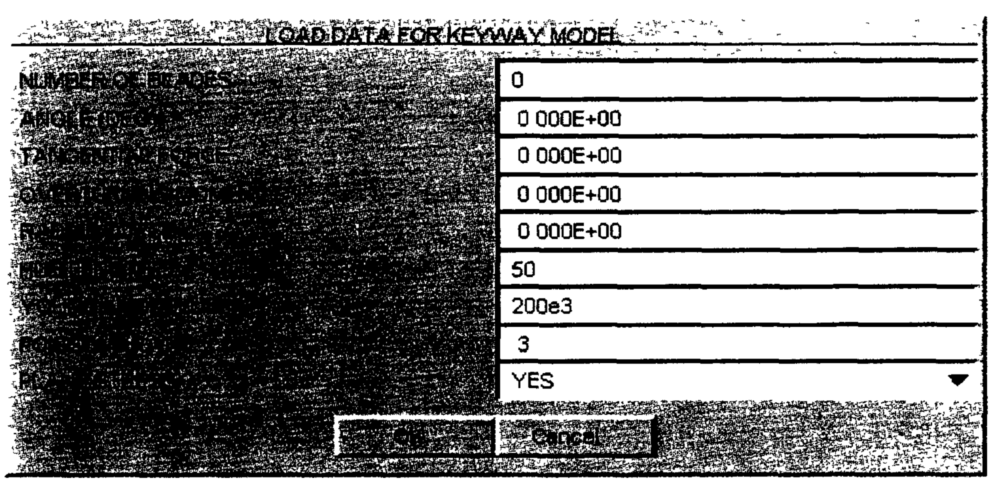


Figure 5.40: Keyway Load Data Submenu

The data entries are defined below:

Number of Blades: Enter the number of blades in the propeller model.

Angle (Deg): Enter the angle in degrees from vertical to spindle axis of first blade (positive clockwise). Angle of first blade must be greater than 90°.

Tangential Force: Enter the tangential force at each blade root (positive clockwise).

Overturning Moment: Enter the overturning moment at each blade root (positive about Z-axis).

Radial Force: Enter the radial force at each blade root (positive outward radial direction).

The other parameters are as defined under the keyway geometry. Click **OK** when finished.

5.14 Air Channel Model

Some propeller blades are provided with an air channel near the blade leading edge. This option is required to model the air bleed channel. The approach adopted is that the blade is first modelled and analyzed, ignoring the air channel effects and then a detailed model of a portion of the blade containing the air channel is generated. The user provides the location of the channel model, as well as the dimensions necessary to generate it. Solid elements are used to generate the air channel model. Figure 5.41 shows the edit table for defining the air channel model.

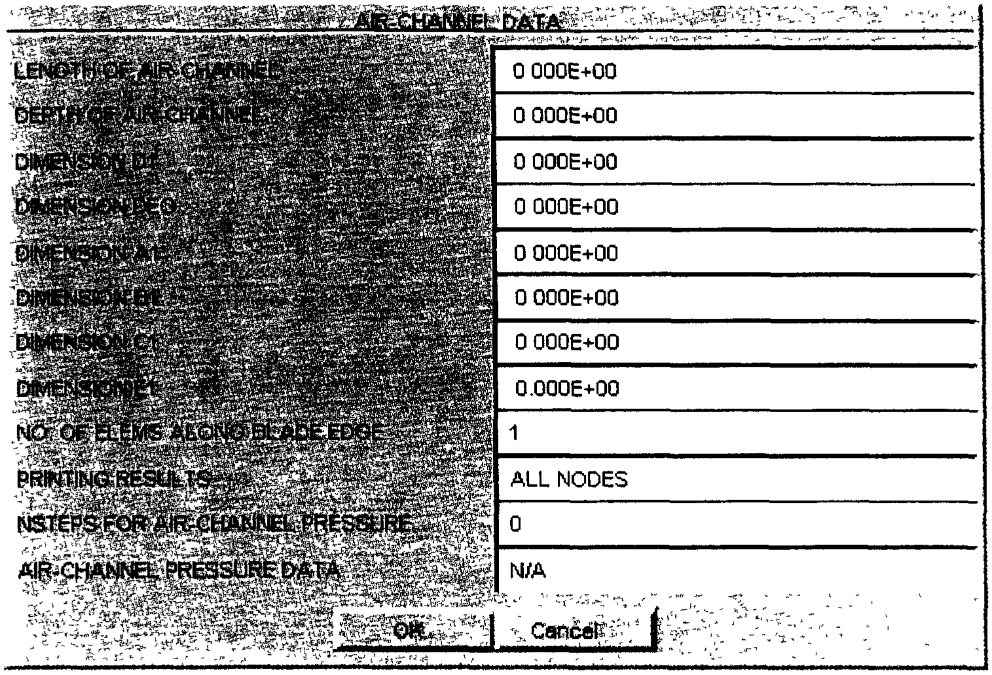


Figure 5.41: Air Channel Data

Refer to Figure 5.42 for illustrations of the dimensions. The parameters are described below:

Length of Air Channel: This is the length of the air channel model along the leading edge (length). Enter the value.

Depth of Air Channel: This is the depth of the air channel perpendicular to the leading edge (length).

Dimension D1: See Figure 5.42. Enter the required value (length).

Dimension DEG: See Figure 5.42. Enter the value in degrees.

Dimension A1: See Figure 5.42. Enter the value (length).

Dimension B1: See Figure 5.42. Enter the value (length).

Dimension C1: See Figure 5.42. Enter the value (length).

Dimension E1: See Figure 5.42. Enter the value (length).

No. of Elems Along Blade Edge: Enter the number of layers of elements along the leading edge.

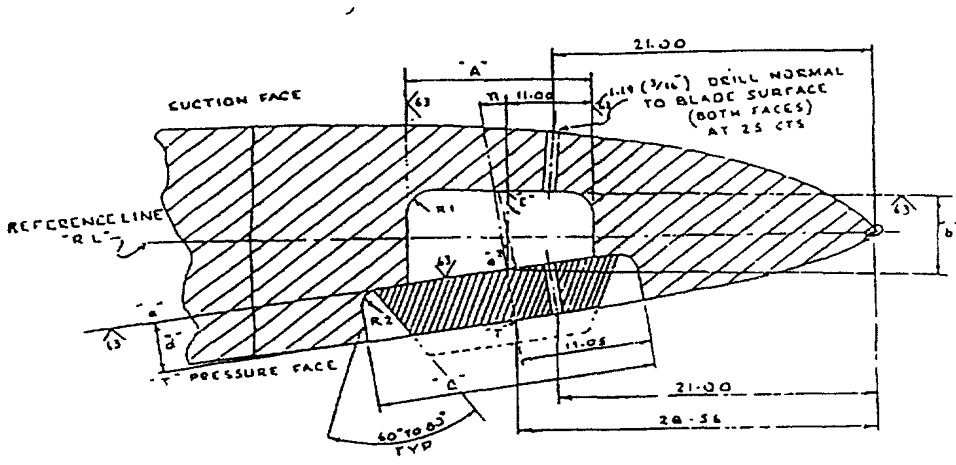
Printing Results: This parameter is used to reduce printing for dynamic analysis. Dynamic results can be printed at **ALL NODES** or at **DEFAULT NODES** computed by the program. Click the appropriate input box to select the required option.

NSTEPS for Air Channel Pressure: Enter the number of steps for which air channel pressure is provided. For static analysis, the number of steps should be set to 1. For dynamic analysis this number should be set to the same value used to describe the dynamic load in the original blade only analysis.

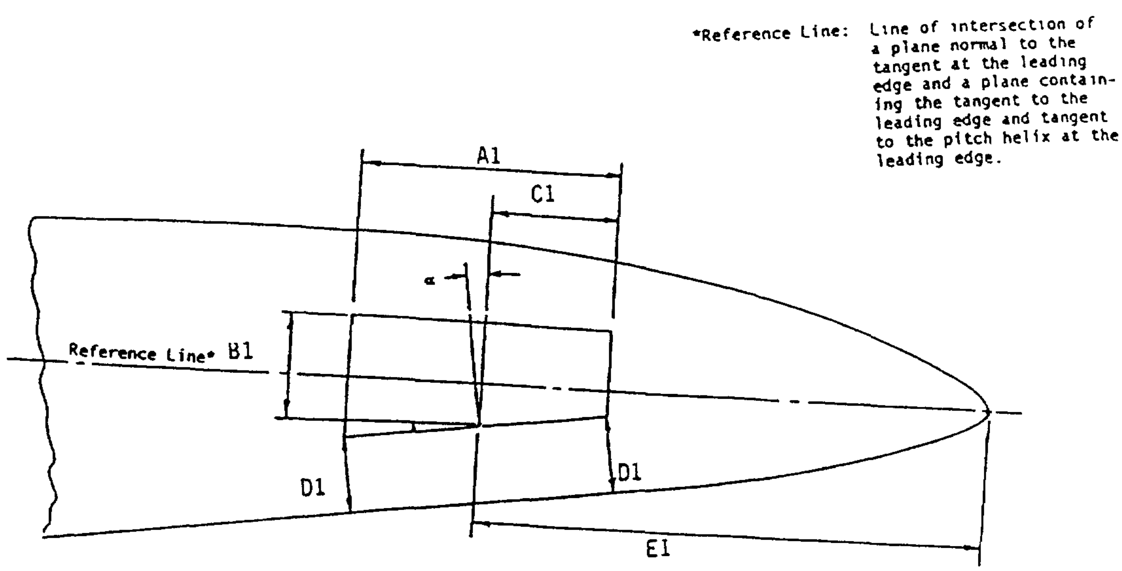
Air Channel Pressure Data: Click on **CREATE** to define the air pressure data at the NSTEPS steps. An edit table will appear for entering the data.

Click **OK** when finished. The next step in the air channel model generation is to select the position of the air channel model. This is carried out interactively.

N.6



(a) Air channel fabrication details



(b) Air channel geometric specifications

Figure 5.42: Propeller Blade Air Channel Specifications

5.15 Properties

This section is intentionally left blank.

6. DISPLAYING PROPELLER BLADE GEOMETRY AND FE MODELS

As discussed in Section 2, PVAST provides two options for displaying the propeller finite element models and input data. These are the *Special* and *General Plotting* capabilities. The Special Plotting options are used to display graphics that are specific to propeller blades, whereas the General Plotting options are used for displaying general purpose finite element models. The display options available under the two display categories are described below. Figure 6.1 shows the **Model Display** submenu, obtained by clicking the **Display** option on the main menu.

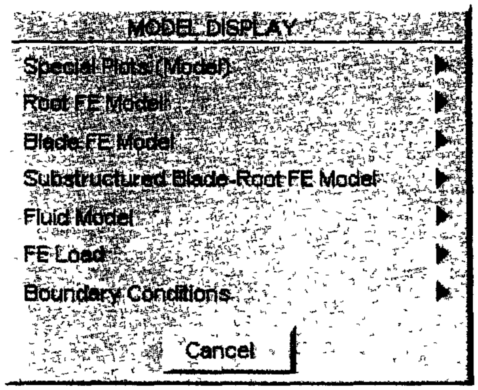


Figure 6.1: Model Display Submenu

The first option on the submenu activates the *Special Plotting* options and all of the remaining options belong to the *General Plotting* option. The display options available under the two display categories are described below.

6.1 Special Plotting Capabilities

Figure 6.2 shows the Special Plotting options.

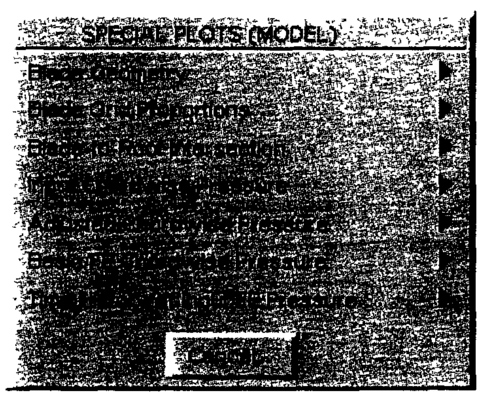


Figure 6.2: Special Plotting Options (Pre-Processing)

The plotting options are defined below.

Blade Geometry: This option displays the blade input data in graphical form. There are several options available under this item, as shown in Figure 6.3.

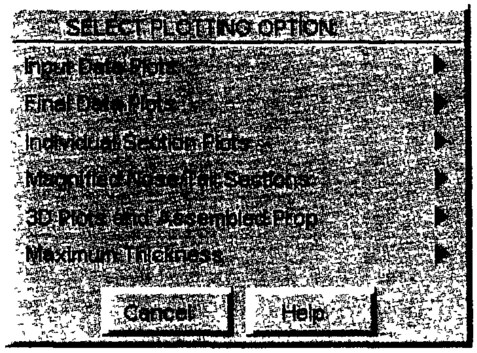


Figure 6.3: Blade Geometry Plotting Options

The **Input Data Plots** and **Final Data Plots** options are used to plot blade data as functions with respect to the radial positions. The data plotted are shown in Figure 6.4.

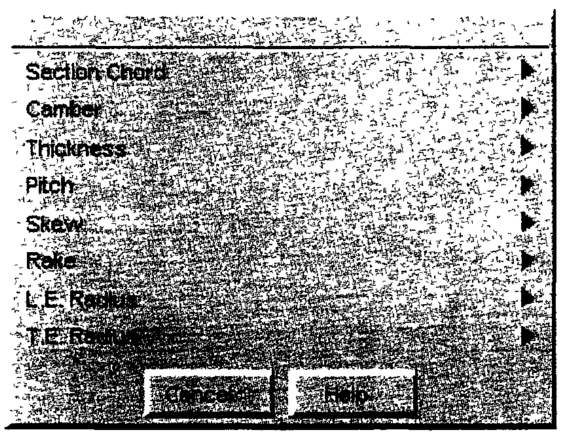


Figure 6.4: Blade Input/Final Data Plot Options

Representative plots the section chord, camber, thickness, pitch, skew, rake, LE radius and TE radius, are shown in Appendix B, Figures B.1-B.8, respectively.

Individual Section Plots are used to plot the blade section at the radial positions. An edit box is provided to enter the section to be plotted. Figure B.9 in Appendix B shows an individual section plot of a typical blade.

Magnified Nose/Tail Sections is used to plot magnified views of blade section at the leading or trailing edges. An edit box is provided to enter the section to be plotted. Figure B.10 in Appendix B shows the magnified noise/tail section of a typical blade.

3D Plots and Assemble Prop is used to provide 3D plots of a single or assembled propeller blades. For single blade, the available options are shown in Figure 6.5.

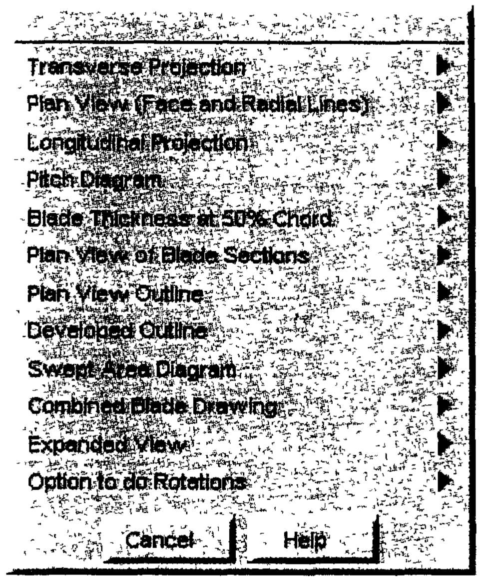


Figure 6.5: 3-D Assembled Propeller Plotting Options

Figures B.11 to B.21 show the views corresponding to all but the last option, respectively, for a typical blade. The last option in Figure 6.5 **Option to do Rotations** is used to define rotation angles. For assembled propeller blades, the available options are shown in Figure 6.6.

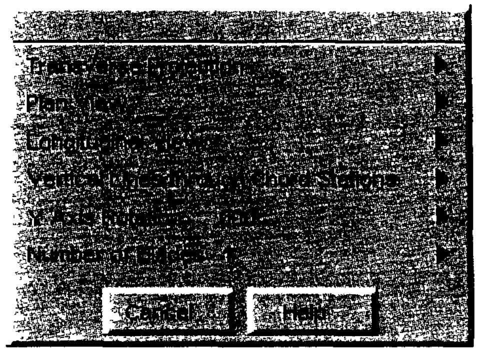


Figure 6.6: Assembled Propeller Blade Plotting

Figures B.22 to B.25 show the views corresponding to the four options: transverse projection, plan view, longitudinal view and vertical lines through chord sections, respectively, for a typical propeller blade.

Maximum Thickness is used to plot the maximum blade thickness along the blade in the radial direction. Options are provided to plot the thickness with or without rake. Figure B.26, in Appendix B shows the maximum thickness with rake, for a typical blade.

Blade Grid Proportions: This option is used to plot the blade finite element grid proportions. Figure B.27 (Appendix B) shows the grid proportions for a typical blade.

Blade-to-Root Intersection: Used to plot the blade-to-root intersection, Figures B.28-B.29 (Appendix B) shows the blade-to-hub and blade-to-palm intersections of typical propeller blades.

Input Chordwise Pressure: Used to plot chordwise variations of the input pressure, available on PREFX.LIN file.

Adjusted Chordwise Pressure: Used to plot the chordwise variations of the adjusted pressure load, available on the PREFX.LAD file.

Basic/FE Chordwise Pressure: Used to plot the chordwise variation of the finite element pressure.

Time History of Input/FE Pressure: Used to plot the time history of the input pressure, for dynamic analysis.

6.2 General Plotting Capabilities (Model)

The general plotting capabilities are used to display finite element models of propeller blade models. Options are provided for displaying the root (fillet, hub or palm), blade, substructured and fluid models as well as the finite element load and boundary conditions. Capabilities are also provided for manipulating the plots (such as changing views, zooming, wire or fill plotting, etc.) as is applicable in the MGDSA general finite element pre-processor [8]. Detailed descriptions of these capabilities are provided

Root FE Model: Plots FE models of the fillet and hub or palm if present. Figure B.30 shows fillet, hub and palm FE models of typical propeller blades, using the fill plotting option.

Blade FE Model: Plots FE models of the propeller blade. Figure B.31 shows a typical blade finite element model.

Substructured Blade-Root FE Model: Plots FE models of substructured blade-root models. Figure B.32 shows typical substructured blade-fillet-hub and blade-fillet-palm models.

Fluid Model: Plots the fluid finite element model if available. For 3D fluid finite elements, the option to plot a thin fluid model (with zero fluid layer thickness) is also provided. Figure B.33 shows typical thin fluid, full 3D fluid finite element and surface panel fluid models.

Boundary Conditions: Displays the applied boundary conditions. A finite element model of the structure should first be displayed prior to selection of this option. Figure B.34 shows a typical blade finite element model with the applied boundary conditions indicated.

7. RUNNING VAST FE ANALYSIS

This option is activated by selecting the **Analysis** option on the main menu. Figure 7.1 shows the submenu that appears with this choice.

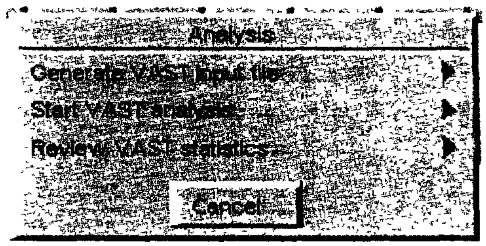


Figure 7.1: Analysis Submenu

The options on this submenu are described below.

Generate VAST Input File: This is used to generate the VAST input file, if not already generated. If the VAST input file PREFX.USE has already been created via Section 5.12, this step is not necessary. Otherwise, generate the input file as described in Section 5.12.

Start VAST Analysis: Select this option to pass the data to the VAST solver to perform a finite element analysis of the propeller blade model. The analysis start up menu will appear as shown in Figure 7.2. Click Start to start the VAST analysis. The status of the analysis will be shown instantaneously, on the start up menu as the analysis progresses. The elapsed time will be shown at the end of the analysis.

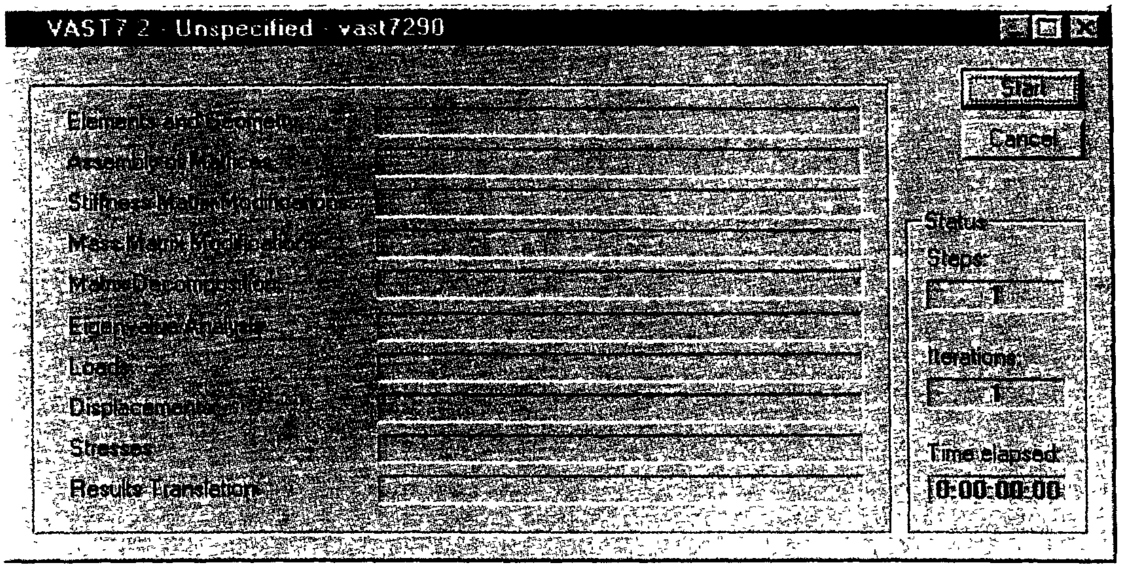


Figure 7.2: VAST Analysis Progress Status

Review VAST Statistics: This option is used to provide a summary of the VAST analysis.

8. DISPLAYING ANALYSIS RESULTS

PVAST provides the capability to display the finite element analysis results graphically. This option is activated by clicking the **Results** option on the main menu. Figure 8.1 shows the **Results** submenu that appears when the option is selected.

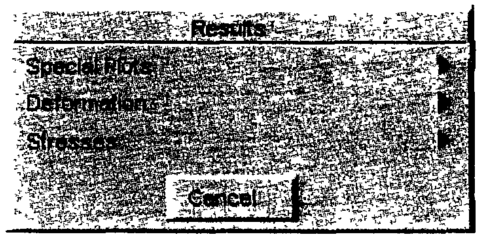


Figure 8.1: Results Submenu

Similar to the model display option, PVAST provides special and general plotting capabilities for displaying the results. The special plotting options can be accessed by clicking on **Special Plots** on the **Results** submenu. The remaining options on the **Results** submenu are all general plotting options. The display options available under the two results display capabilities are discussed below.

8.1 Special Plotting Capabilities

Figure 8.2 shows the special results display options. It should be noted that this option is not available when a blade with an optimized tip is used.

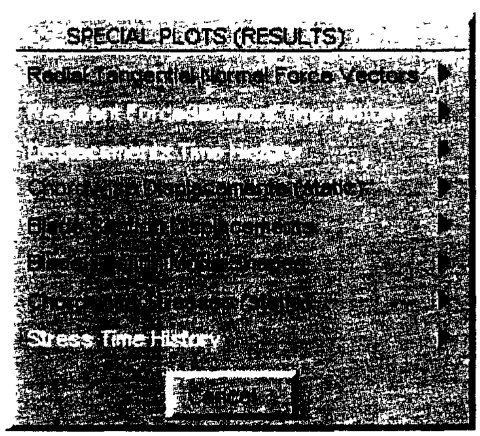


Figure 8.2: Results Special Plotting Options (Results)

The figure shows all the available options. However, options that are not applicable for a particular analysis are grayed out as shown. The plotting options are defined below.

Radial, Tangential, Normal Force Vectors:

This option is used to display the radial, tangential and normal force vectors. An option to plot all force components on the same plot is also provided. The option plotted is obtained by toggling through the available options: **Radial**, **Tangential**, **Normal** and **Combined** as shown in Figure 8.3.

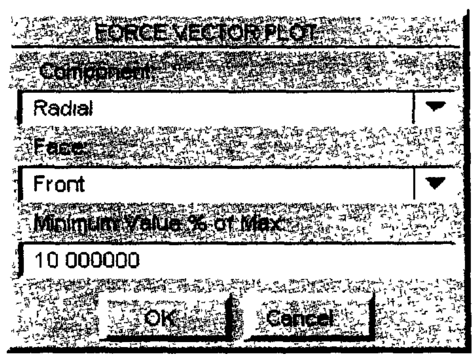


Figure 8.3: Submenu for Displaying Force Vectors

If the plot is too clustered, it may be desirable to reduce the number of vectors plotted by specifying the minimum size of vector as a percentage of the maximum. Enter the desired percentage in the box shown in Figure 8.3. Typical radial, tangential normal and combined force vectors are shown in Figure C1-C4 of Appendix C.

Resultant Force and Moment Time History:

This option is used to plot the resultant force and moment time histories from dynamic transient analysis. Typical resultant force and moment time histories are shown in Figures C5-C6 of Appendix C.

Displacement Time History:

This option is used to plot displacement time histories at selected radial and chordwise position. The user is required to select the radial and chordwise position at which the displacement history is to be plotted. A typical displacement time history plot is shown in Figure C7 in Appendix C.

Chordwise Displacements (Static):

This option is used to plot the chordwise variation of static displacement results at selected radial positions. A list of the radial positions will be presented and the user can select the desired radial position by clicking on the appropriate position.

The option to plot the radial, tangential or normal components of the displacement is provided. Select the desired component by toggling through the options: **Normal, Tangential** and **Radial**. Figures C8-C10 (Appendix C) show typical chordwise plots of the normal, tangential and radial displacement components, respectively.

Blade Section Displacements:

This option is used to display the blade section displacements. A list of the radial positions will be presented and the user can select the desired blade section by clicking in the appropriate radial position. PVAST displays the deformed and underformed blade section.

A magnification factor for the displacements can also be entered to clearly highlight the deformation.

For transient analysis, the time step at which the displacement is to be plotted is also required. Figure C11 of Appendix C, shows a typical blade section displacement plot.

Blade Section Mode Shapes:

This option is similar to the **Blade Section Displacement** plot option, except that mode shapes are plotted here instead of static or transient displacements.

In addition to specifying the radial position and magnification factor as described above, the user is required to enter the mode to be plotted. Figure C12 (Appendix C) shows typical blade section mode shapes.

Chordwise Stresses (Static): This option is similar to the **Chordwise Displacements (Static)** option, except that stresses rather than displacements are plotted here. The stress components available for plotting are the **Primary Principal, Secondary Principal, Tangential** and **Radial** stress components.

Figures C14-C17 of Appendix C, show typical chordwise plots of the primary principal, secondary principal, tangential and radial stress components.

Stress Time History: This option is used to plot stress time histories from transient dynamic analysis, similar to the **Displacement Time History** option discussed above.

The user is required to select the stress component and the radial and chordwise fraction of the point at which the stress history is to be plotted.

Figure C17 of Appendix C shows stress time history plots of a typical propeller blade.

8.2 General Plotting Capabilities (Results)

The general plotting capabilities are used for displaying the finite element displacements, mode shapes and stress results as in the MGDSA [8] general purpose finite element pre- and post-processing program. Capabilities for manipulating the plots (such as changing view, zooming, wire or fill plotting, etc.) are provided. Prior to plotting the results (via the general plotting capabilities) it is necessary to display the finite element model of the propeller blade or substructure, as discussed in Reference [8]. The available plotting options are discussed below:

Deformation: This option is used to plot the deformations from static or dynamic analysis or vibration modes. The user responses are very straightforward.

Figure C18 (Appendix C) shows a typical deformation plot for static analysis and Figure C19 a typical mode shape from a vibration analysis.

To plot displacement contours use the following steps from the main menu:
Options→Plotting→Contour→Stresses

Then select fringe or line plotting.

Stresses:

This option is used to plot the static or transient dynamic stresses as in the MGDSA general purpose finite element pre- and post-processing program. This option is activated by clicking the **Stresses** option in the **Results** submenu.

The user responses are described fully in Reference [8]. Figure C20 (Appendix C) shows typical element stress plots.

To plot stress contours use the following steps from the main menu:
Options→Plotting→Contour→Stresses
Then select fringe or line plotting.

9. TOOLS

This option is activated by clicking on **Tools** on the PVAST main menu. The tools provide a means of manipulating the general purpose model and results display capabilities. Figure 9.1 shows the Tools submenu. The four options available are summarized in Section 9.1 to 9.4, respectively. Detailed descriptions are provided elsewhere [8].

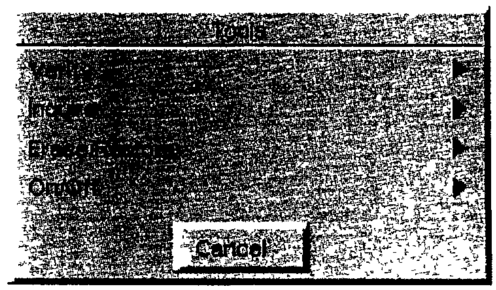


Figure 9.1: Tools Submenu

9.1 Model Verification

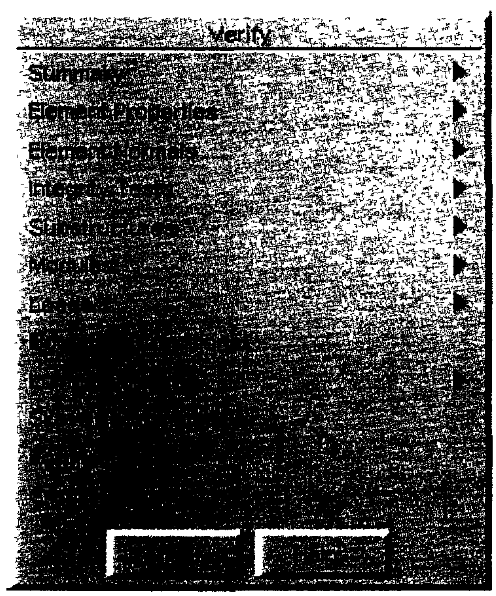


Figure 9.2: Model Verification Items

Figure 9.2 shows the model verification submenu. The items on the menu are summarized below.

Summary:	Provides a summary of the finite element model (eg. summary of nodes, elements, boundary conditions, etc.)
Element Properties:	Used to verify properties of the elements.
Element Normals:	Plots element normals.
Integrity Test:	Performs integrity tests on the model (eg. element aspect ratios, element angles, etc.).
Substructures:	For verifying substructures.
Modules:	Not applicable for PVAST.
Loads:	For verifying loads.
Boundary Conditions:	For verifying boundary conditions.
Lumped Masses:	For verifying lumped masses.
Skew Coordinates:	For viewing the skew coordinates if available.
Formulation:	For verifying the FE formulations (eg. linear elastic, nonlinear, etc.).
Crack Tips:	Not applicable to PVAST.

9.2 Inquiry Operations on FE Models

This option is sued to perform inquiries on the finite element model. The items for which inquiries can be made include nodes, elements, properties, grid, pressures, forces/moments, deformations and stresses. These are provided in the **Inquiry** submenu shown in Figure 9.3. Detailed description of the inquiry on the items is provided in reference [8].

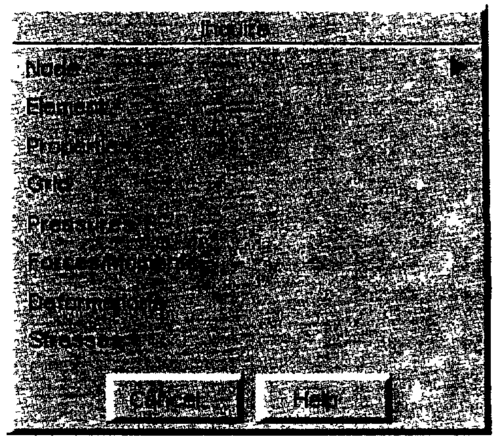


Figure 9.3: Inquiry Submenu

9.3 Erasing and Restoring Models

This option is used to erase or restore various parts of the finite element model as a convenient way of manipulating the model for a desired effect. Figure 9.4 shows the **Erase/Restore** submenu and the available items are summarized below.

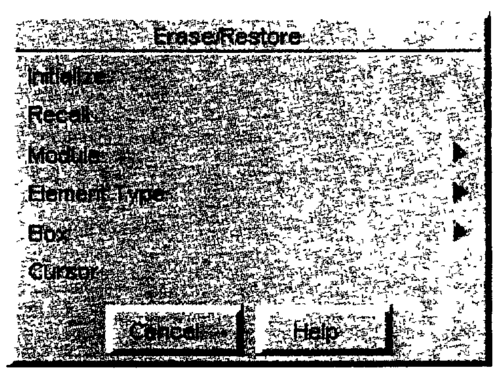


Figure 9.4: Erase/Restore Submenu

Initialize Recall
Module: Used to recall previously displayed model.

Element Type
Box: Enables selection of element types to be erased or enables user to define a boxed area such that elements within the box can be erased or restored.

Cursor: Enables cursor selection to select elements or other finite element model components to be erased or restored.

9.4 On/Off Switches

This option is used to activate the On/Off switches for various general purpose model and results display capabilities. The items available are provided in the **On/Off** submenu shown in Figure 9.5.

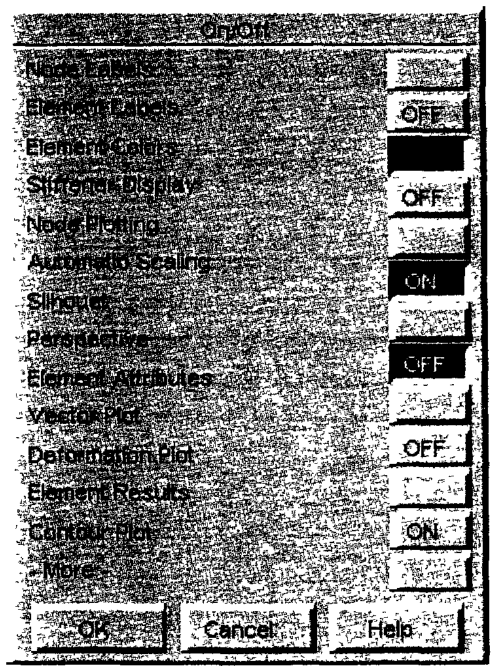


Figure 9.5: On/Off Subfiles

The various items are turned on or off by clicking the **On/Off** button beside the items. Further descriptions of these capabilities are provided in reference [8].

10. OPTIONS

This option is activated by clicking **Options** on the PVAST main menu. Figure 10.1 shows the items available on the Options submenu and they are summarized below.

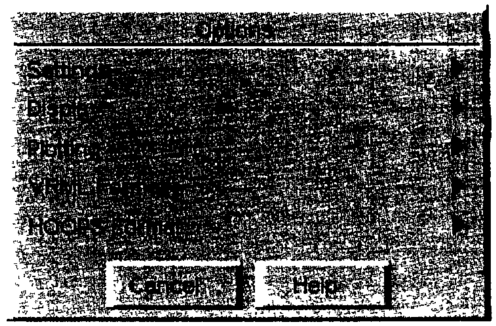


Figure 10.1: Options Submenu

Settings: Used to set the background colour, shrink factors, zooming methods, etc. The complete list of items on the **Settings** submenu is shown in Figure 10.2.

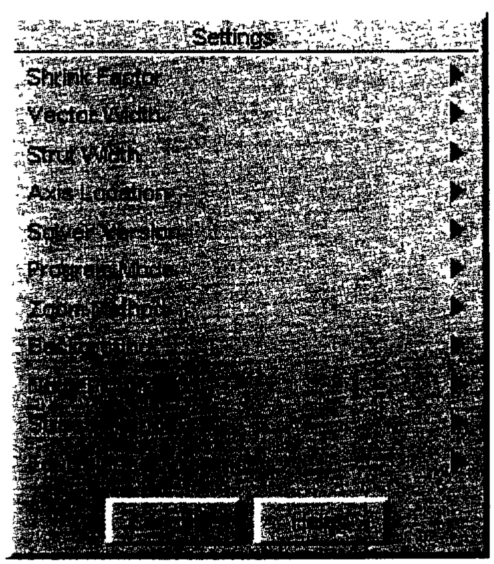


Figure 10.2: Settings Submenu

Display: Used to set display properties (such as viewing, zooming, panning, etc.). The complete list of items on the graphics display submenu is provided in Figure 10.3.

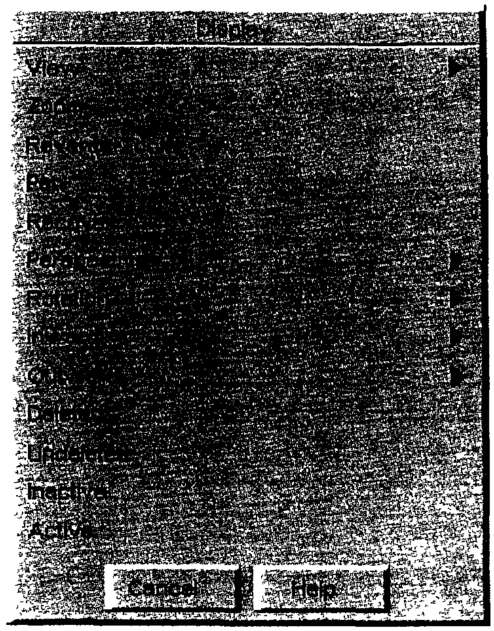


Figure 10.3: Graphics **Display** Submenu

Plotting: Used to set plotting options, such as wire mesh, hidden line, contours, etc. The complete list of items available in the plotting submenu is shown in Figure 10.4.

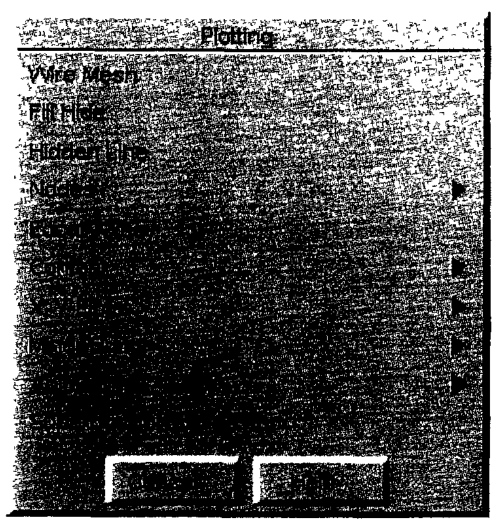


Figure 10.4: Plotting Submenu

VRML Format: Not applicable for PVAST.

HOOPS Format: Not applicable for PVAST.

Further details are available in reference [8].

11. REFERENCES

[1] M.E. Norwood. "A Digital Computer Program for the Vibration and Stress Analysis of Propellers – User's Manual." Canplan Oceanology Limited, February 1975.

[2] D.R. Smith and T.R. MacFarlane. "BLADE: A Propeller Blade Geometry Generator Program User's Manual." DREA Technical Communication 91/302, February 1991.

[3] J.L. Kerwin and C.S. Lee. "Prediction of Steady and Unsteady Marine Propeller Performance by Numerical Lifting-Surface Theory." SNAME Trans., Vol. 86, 1978.

[4] M.E. Norwood. "Propeller Vibration and Stress Analysis by the Finite Element Methods (PVAST), User's Manual." Martec Limited, Halifax, 1980.

[5] M.W. Chernuka, M.E. Norwood and T.S. Koko. "Propeller Vibration and Stress Analysis by Finite Element Methods (PVAST) Version 4 User's Manual." Martec Limited, July 1992.

[6] M.W. Chernuka, M.E. Norwood and T.S. Koko. "Propeller Vibration and Stress Analysis by Finite Element Methods (PVAST) Version 6 User's Manual." Martec Limited, July 1993.

[7] Vibration and Strength Analysis Program (VAST): User's Manual Version 7.3, Martec Limited, April 1997.

[8] MicroVAST Graphics System User's Manual. Martec Limited, March 1996.

APPENDIX A: FORMAT OF PREFIX.LIN FILE CONTAINING HYDRODYNAMIC LOAD ON PROPELLER BLADE

Card 1 (I5)

NDATA = Number of blade loading data sets.

Repeat Cards 2-12 for each data set

Card 2 (3I5)

IPRESS = Code for specifying type of pressure loading. If = 1, a uniform pressure loading is generated using a specified face pressure and a specified back pressure. If = 2, pressures at specified radial and chordwise locations are provided, thus permitting non-uniform pressure loading.

ILTYPE = Code describing the type of loading. If = 1, the loading is considered to be static. If = 2, the loading is considered to be dynamic. If = 3, the dynamic loading is considered to be steady periodic. The user describes the loading function for one complete period including end points. The desired number of repeating periods will be generated automatically.

ILPER = Total number of periods for dynamic loading function. Set zero if ILTYPE ≠ 3.

Card 3 (2I5) (Omit if IPRESS=1)

K = Number of radial positions where pressures are specified (≥2).

L = Number of chordwise positions where pressures are specified (≥2).

Card 4 (8F10.4) (Omit if IPRESS=1)

$R_{(1)}$ = The fraction radii where pressures are specified.
↓
$R_{(K)}$

Card 5 (8F10.4) (Omit if IPRESS=1)

$C_{(1)}$ = The fraction chords where pressures are specified.
↓
$C_{(L)}$

Card 6 (I5) (Omit if ILTYPE=1)

NLV = Number of points in time; i.e. the number of pressure values used to describe dynamic load history.

Provide Cards 7-10 for NLV points in time.

Card 7 (E10.3) (Omit if ILTYPE=1)

TIME = Time in seconds.

Card 8 (2F10.4) (Omit if IPRESS=2)

UPB = Uniform pressure on back (force/length^2). (Positive towards surface.)

UPF = Uniform pressure on face (force/length^2). (Positive towards surface.)

Repeat Cards 9 and 10 for I=1,K

Card 9 (PB_{IJ}, j=1,L) (8F10.4) (Omit if IPRESS=1)

$PB_{(I,1)}$ ↓ $PB_{(I,L)}$ = Pressure loading on back (force/length^2). (Positive towards surface.)

Card 10 (PF_{IJ}, j=1,L) (8F10.4) (Omit if IPRESS=1)

$PF_{(I,1)}$ ↓ $PF_{(I,L)}$ = Pressure loading on face (force/length^2).

Card 11 (I5)

NDF = Number of radial positions where lumped drag forces at midchord are specified.

Card 12 (2F10.4) (Omit if NDF=0)

RD_I = The fraction radii where drag forces are specified.

DF_I = Drag force.

Repeat Card 12 for I=1,NDF Repeat Card 12 for NLV points in time

APPENDIX B: SAMPLE PLOTS OF PROPELLER BLADE GEOMETRY AND FE MODELS

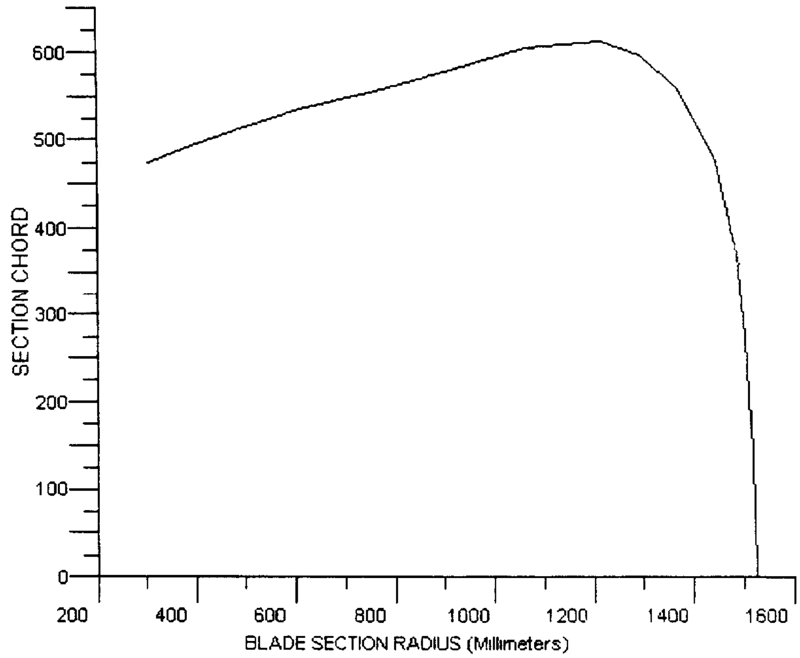


Figure B.1: Section Chord Versus Section Radius

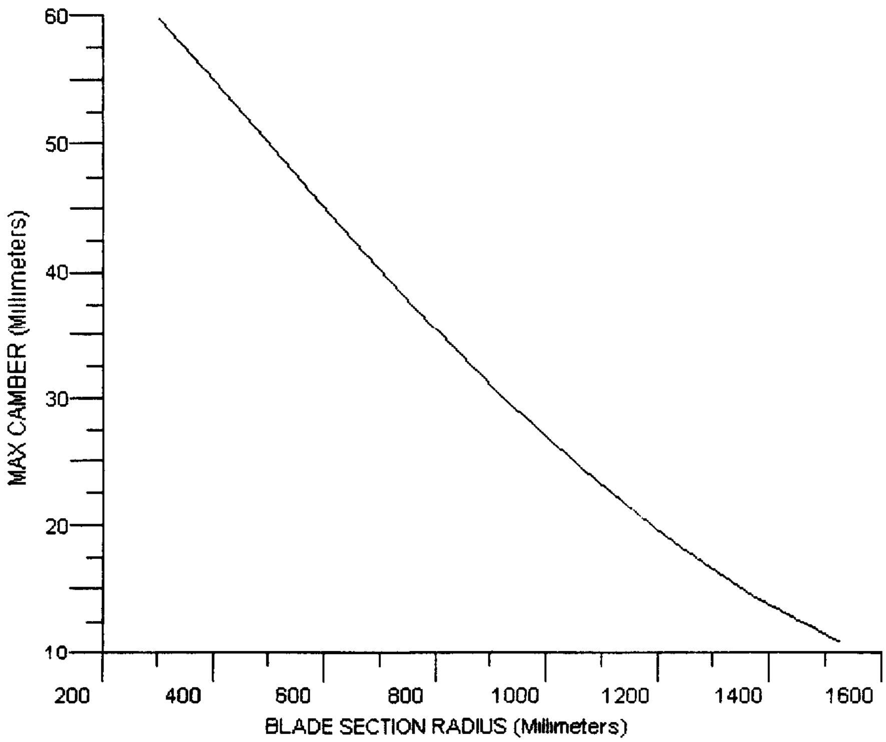


Figure B.2: Maximum Camber Versus Section Radius

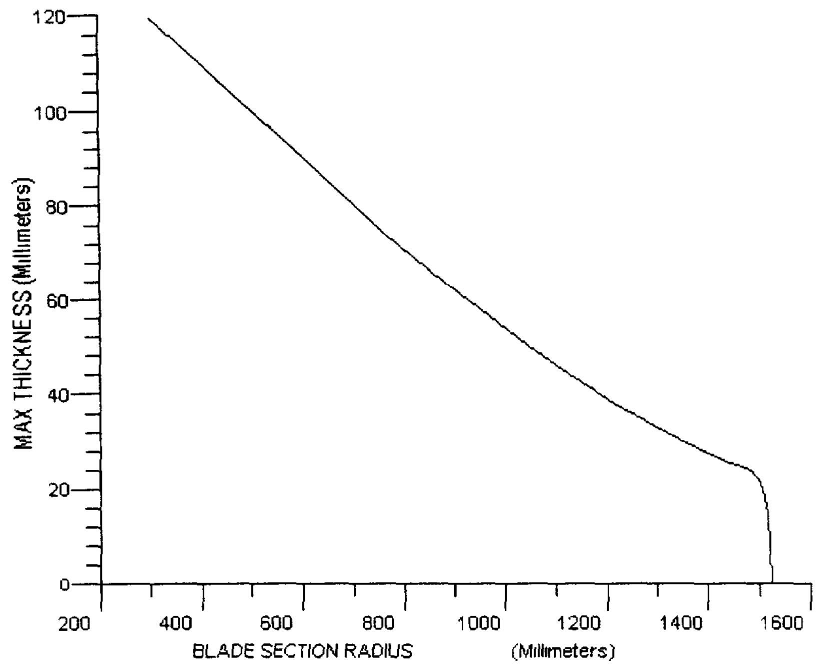


Figure B.3: Maximum Thickness Versus Section Radius

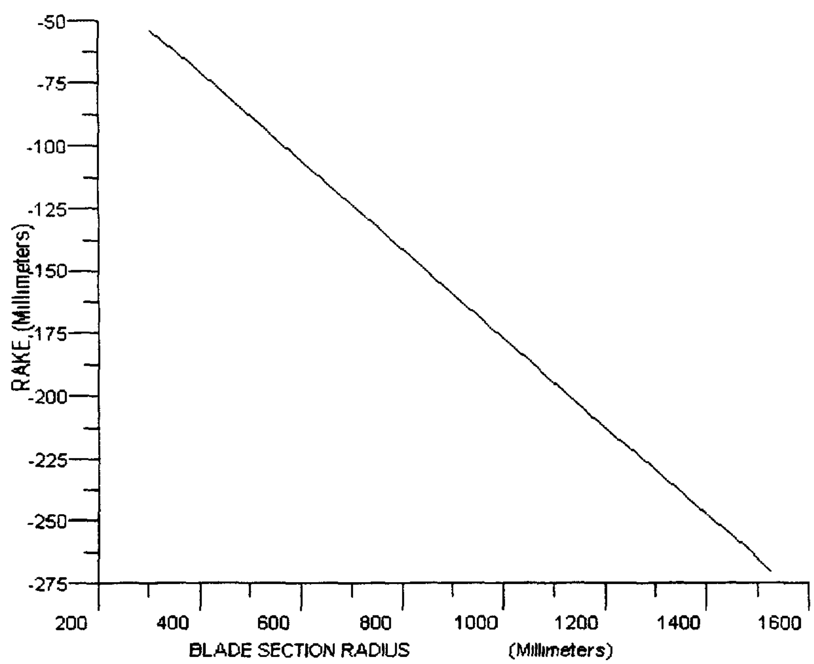


Figure B.4: Pitch Version Section Radius

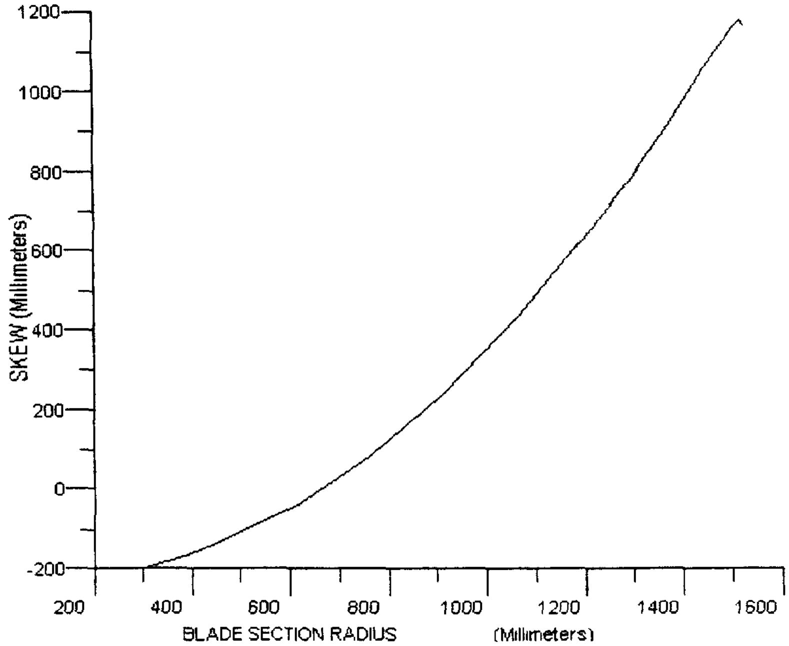


Figure B.5: Skew Versus Section Radius

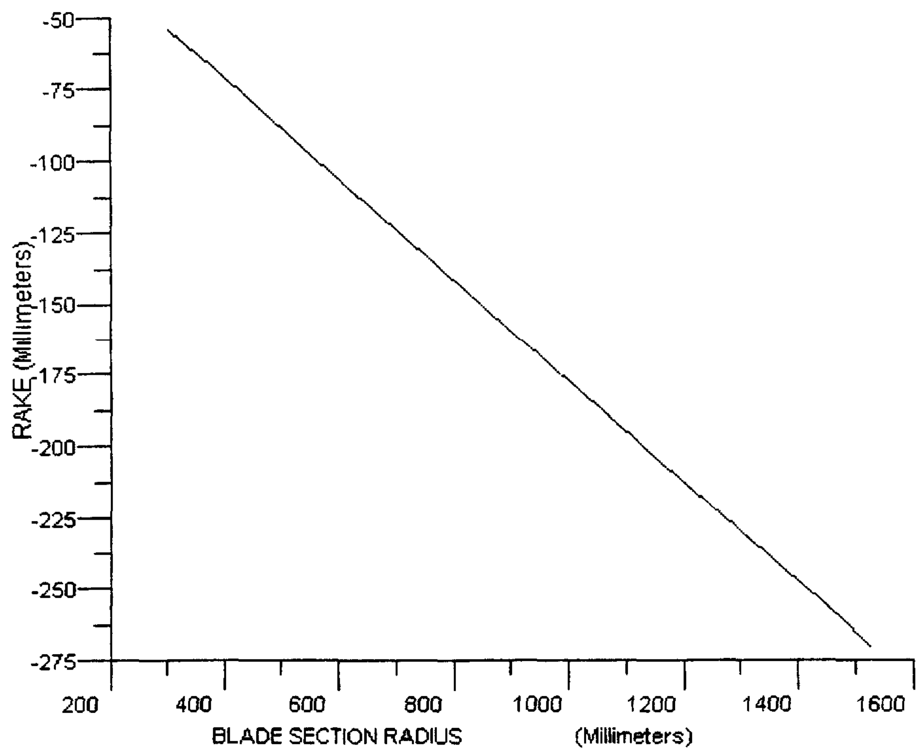


Figure B.6: Rake Versus Section Radius

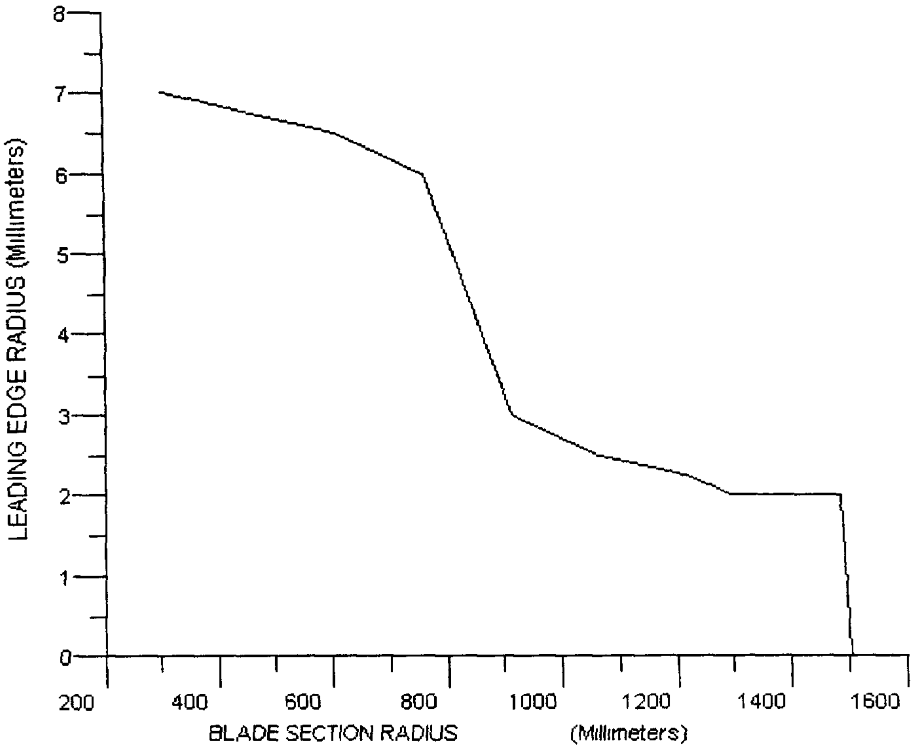
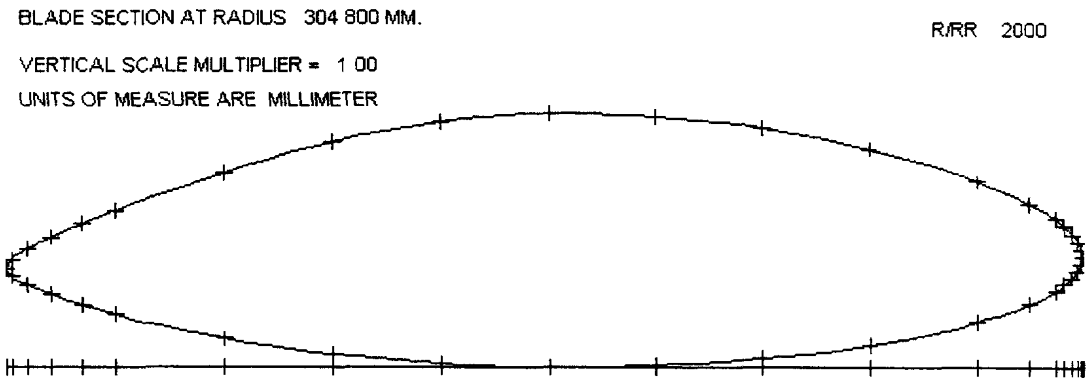


Figure B.7: LE Radius Versus Section Radius

Figure B.8: TE Radius Versus Section Radius



CHORD	475 0	435 8	409 3	380 0	284 9	189 9	94 8	2	95 2	190 1	285 1	380 0	427 5	451 3	475 0
BACK	-46 4	-61 0	-67 2	-73 8	-91 5	-105 7	-115 5	-119 6	-117 9	-112 8	-102 9	-87 6	-77 0	-69 7	-52 3
FACE	-46 4	-33 9	-29 2	-24 5	-13 3	-6 1	-1 7	0	- 9	-4 0	-10 2	-21 2	-29 7	-35 9	-52 3

Figure B.9: Individual Section Plot

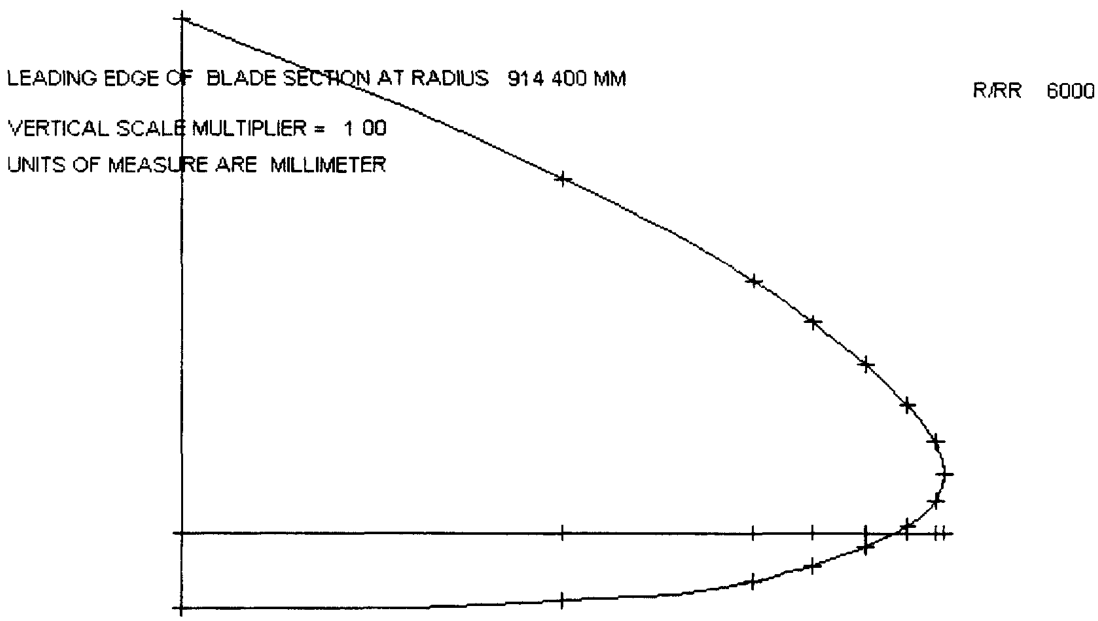


Figure B.10: Magnified Noise/Tail Section

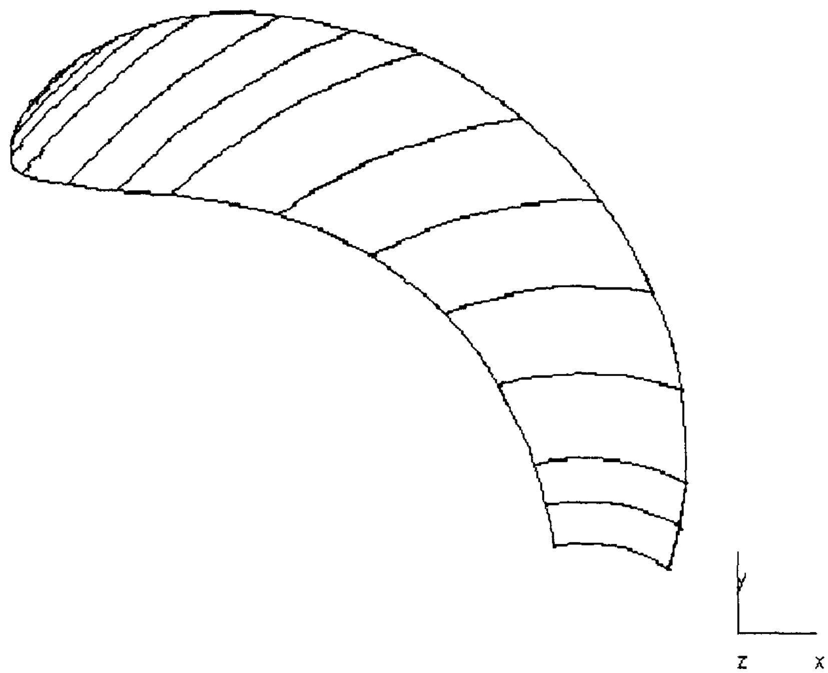


Figure B.11: Transverse Projection

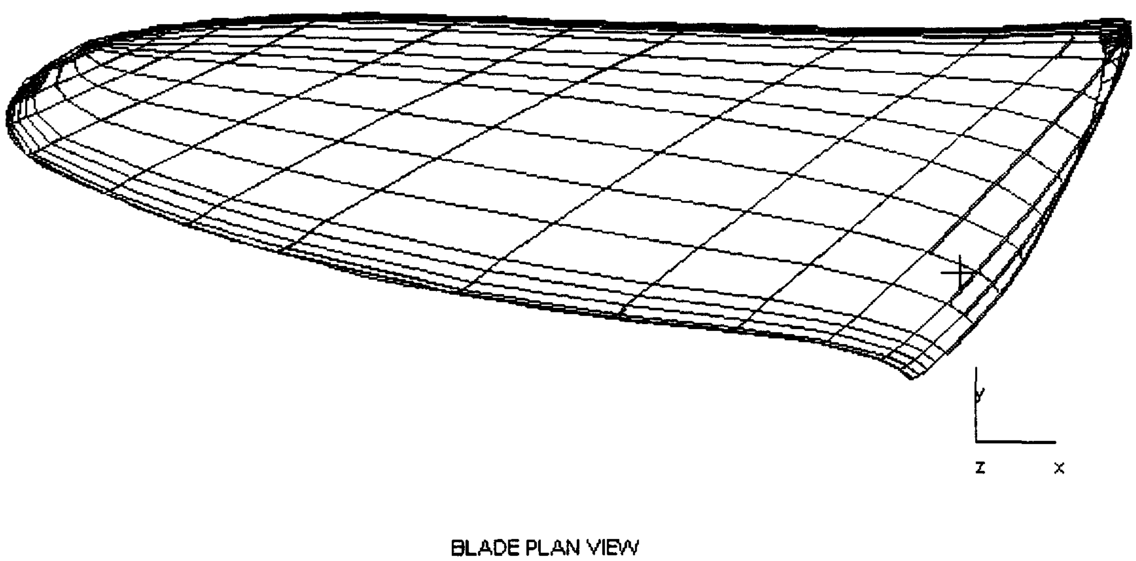


Figure B.12: Plan View

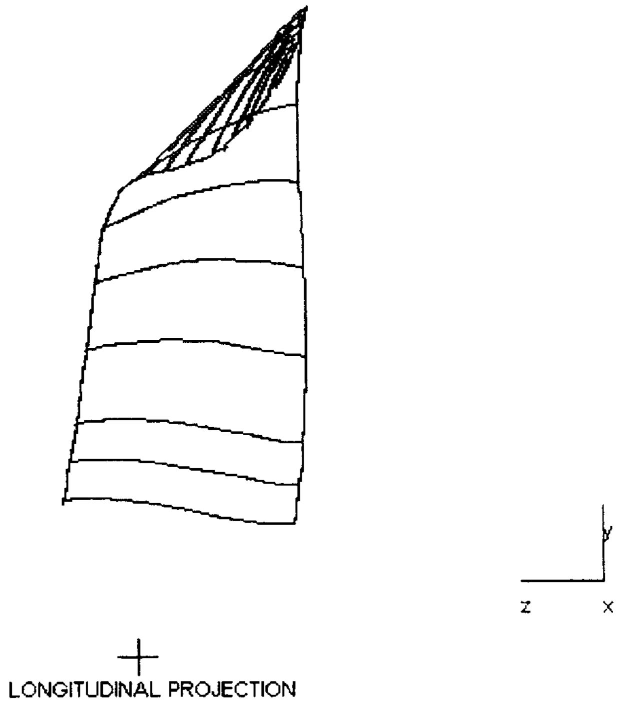


Figure B.13: Longitudinal Projection

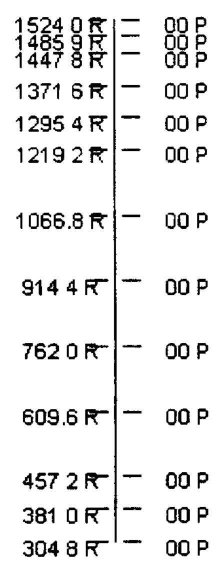


Figure B.14: Pitch Diagram

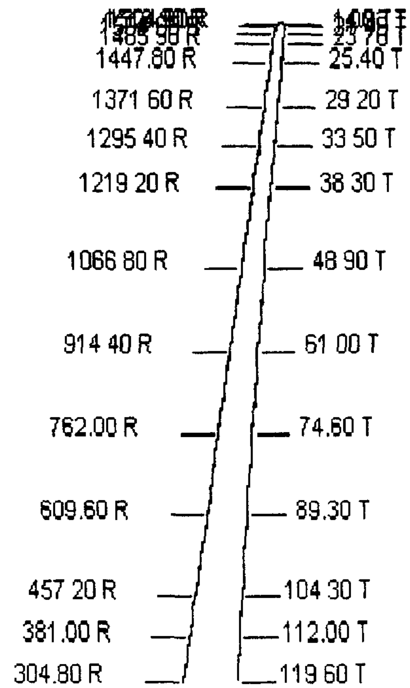


Figure B.15: Blade Thickness at 50% Chord

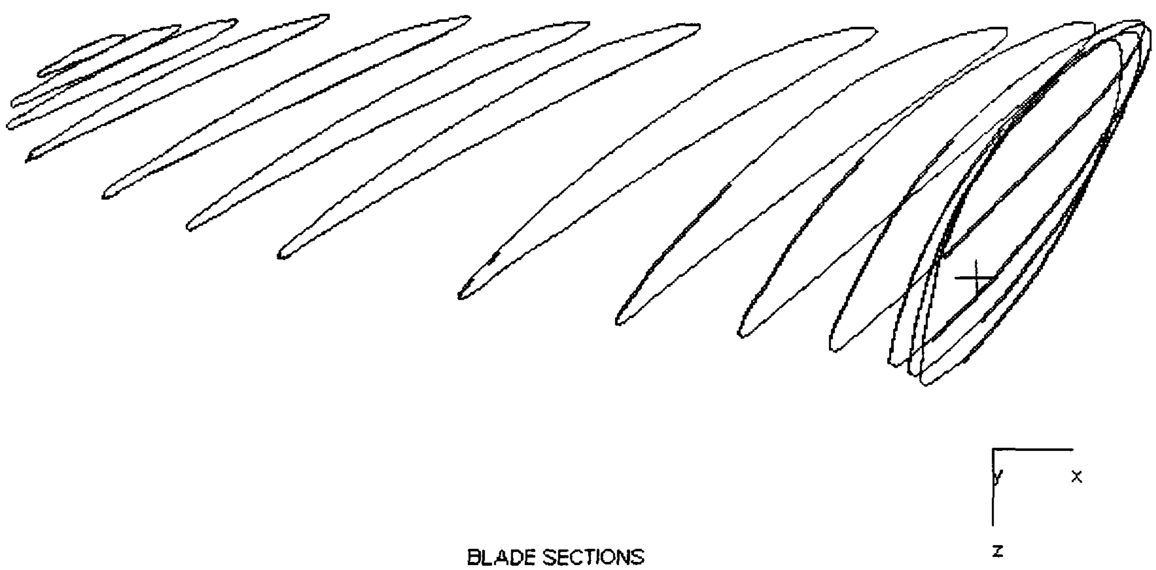


Figure B.16: Plan View of Blade Sections

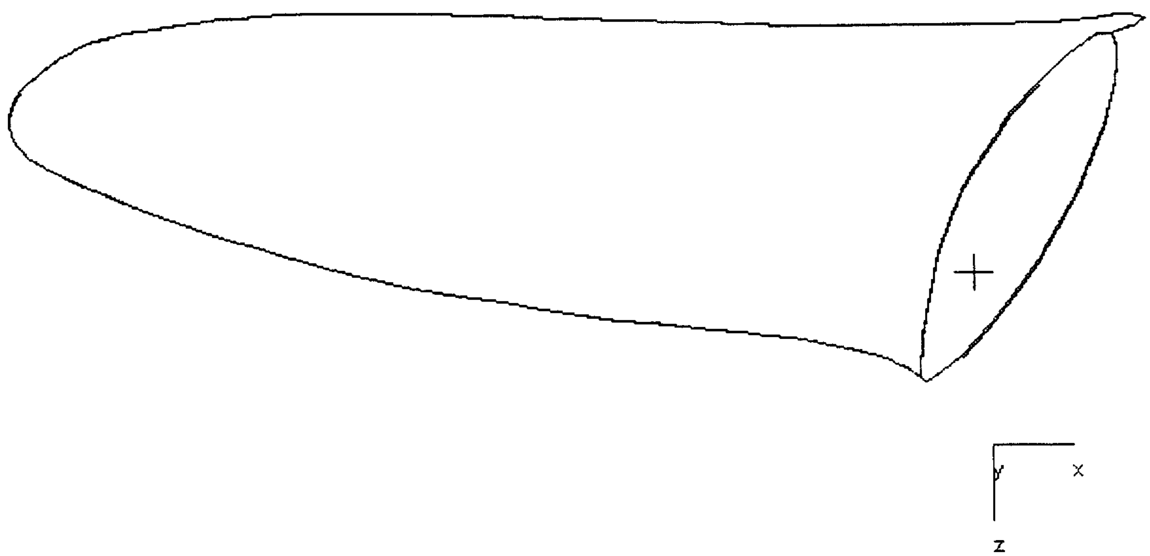


Figure B.17: Plan View Outline

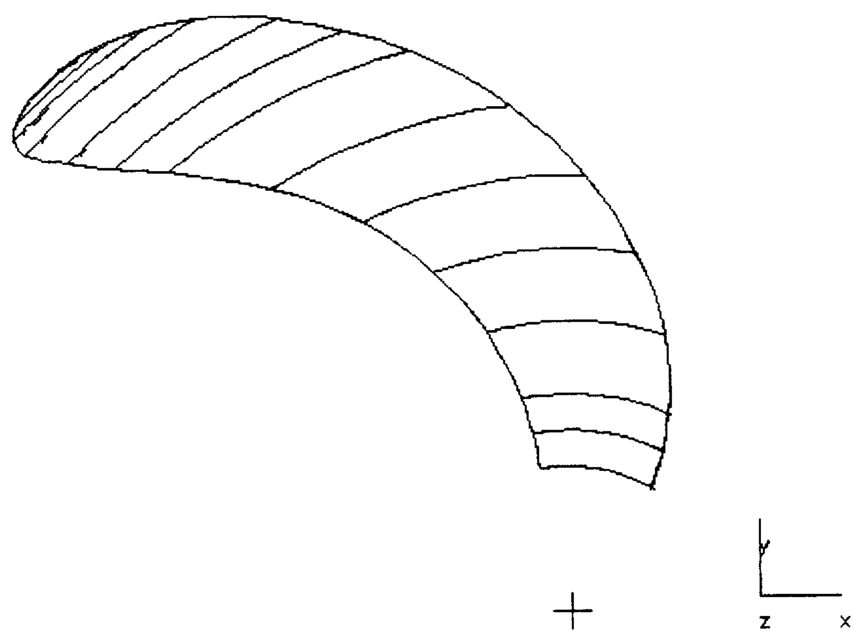


Figure B.18: Developed Outline

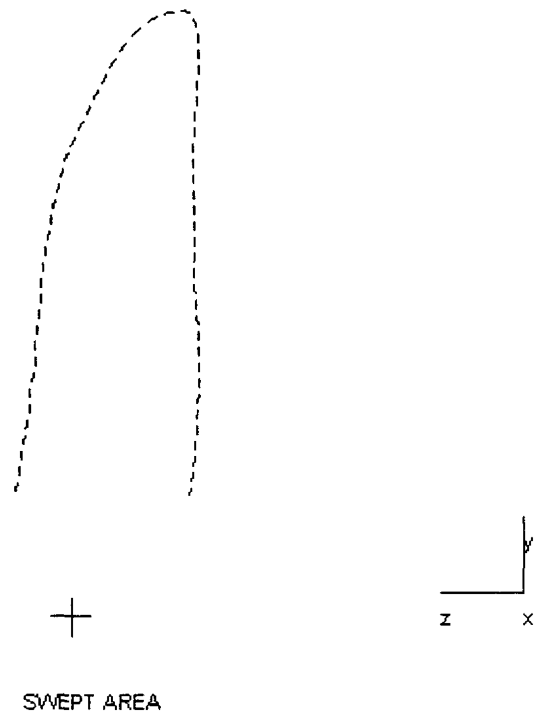


Figure B.19: Swept Area Diagram

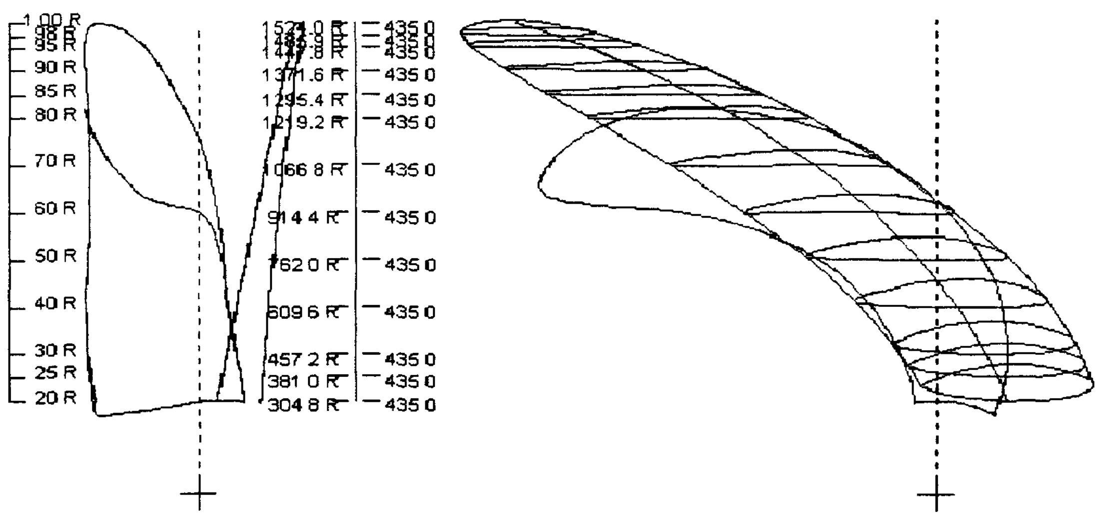


Figure B.20: Combined Blade Drawing

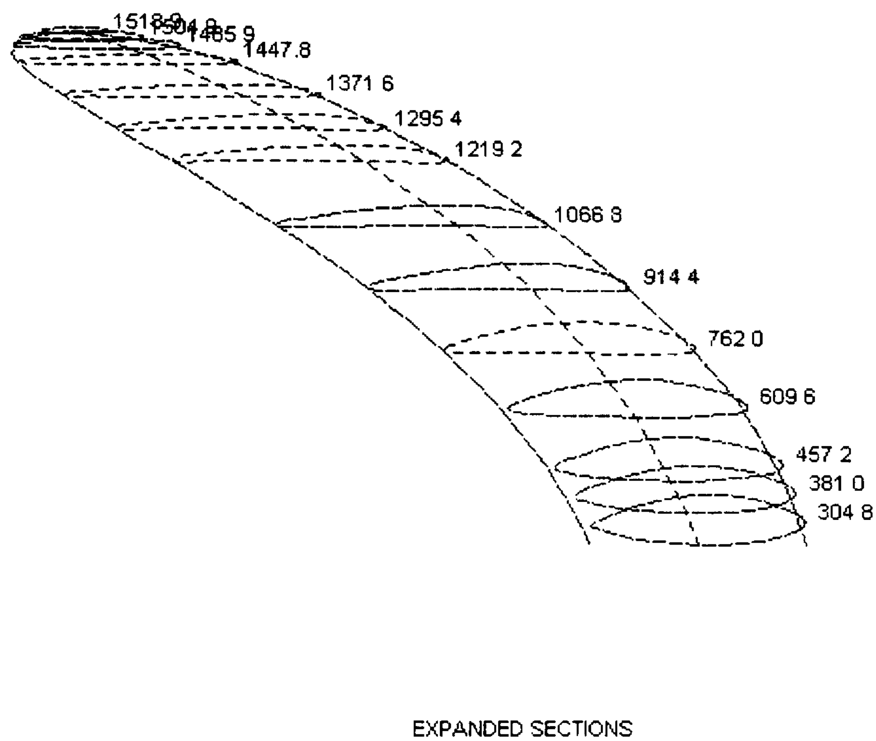


Figure B.21: Expanded View

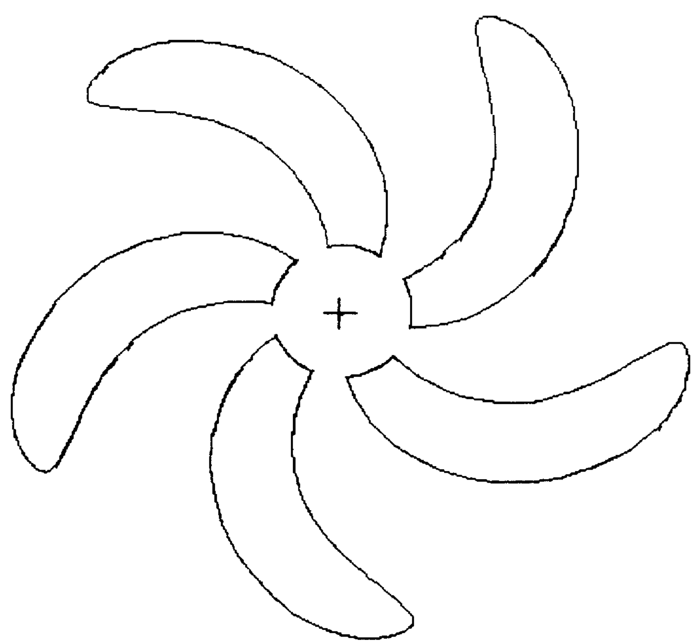


Figure B.22 Transverse Projection of Assembled Propeller Blade

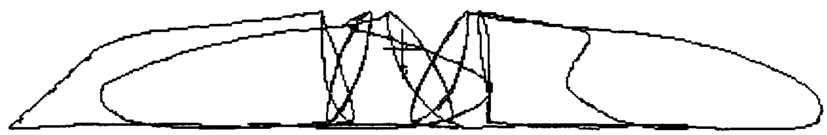


Figure B.23: Plan View of Assembled Propeller Blade

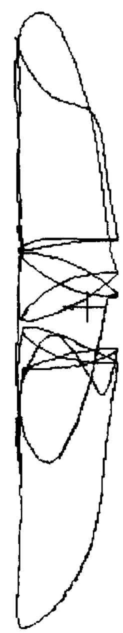


Figure B.24: Longitudinal View of Assembled Propeller Blade

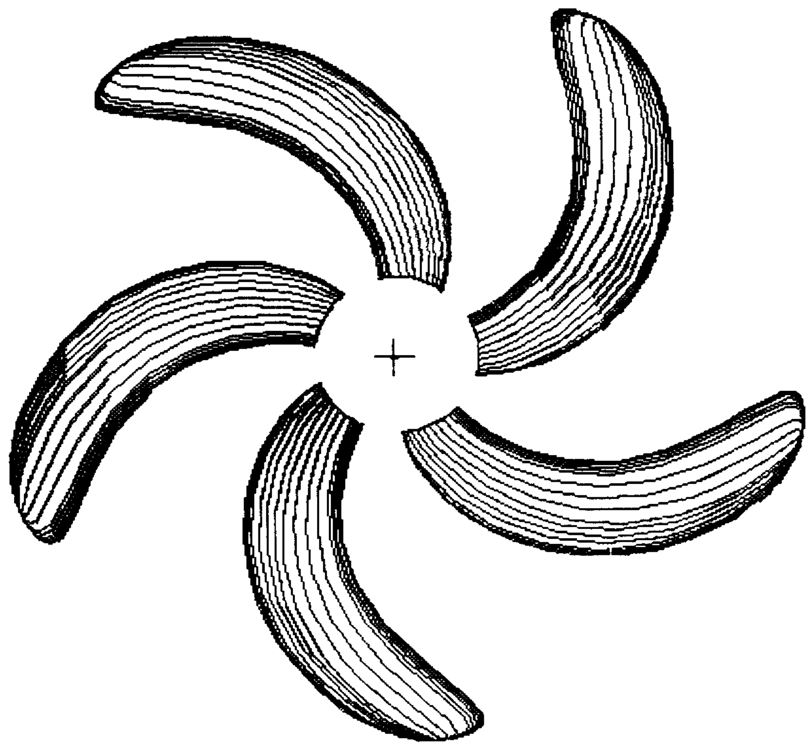
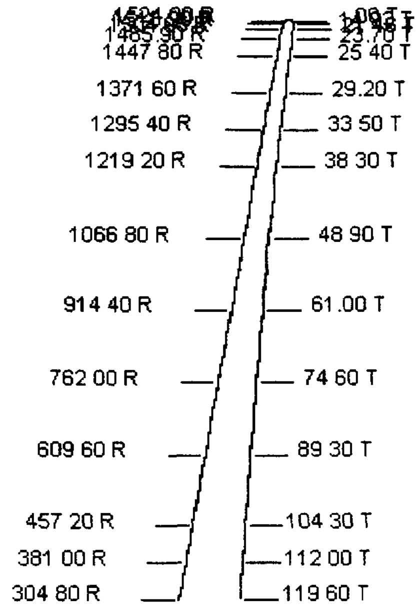
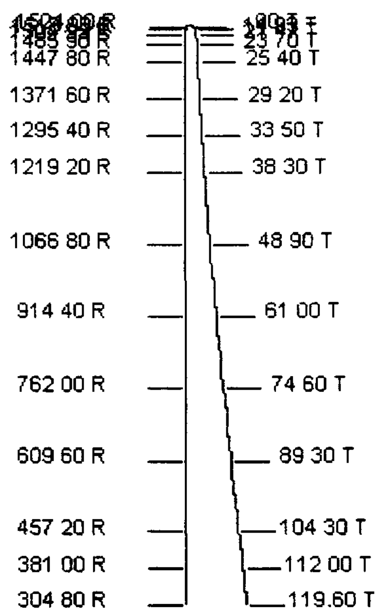


Figure B.25: View of Assembled Propeller Blade with Vertical Lines Through Chord Sections



(a)



(b)

Figure B.26: Plots of Maximum Thickness: (a) With Rake; (b) Without Rake

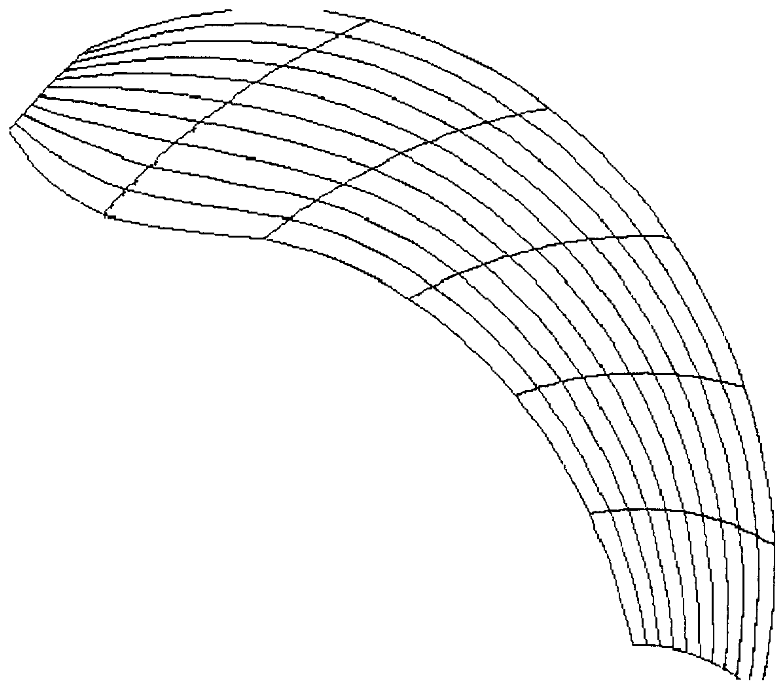


Figure B.27: Blade Finite Element Grid Proportions

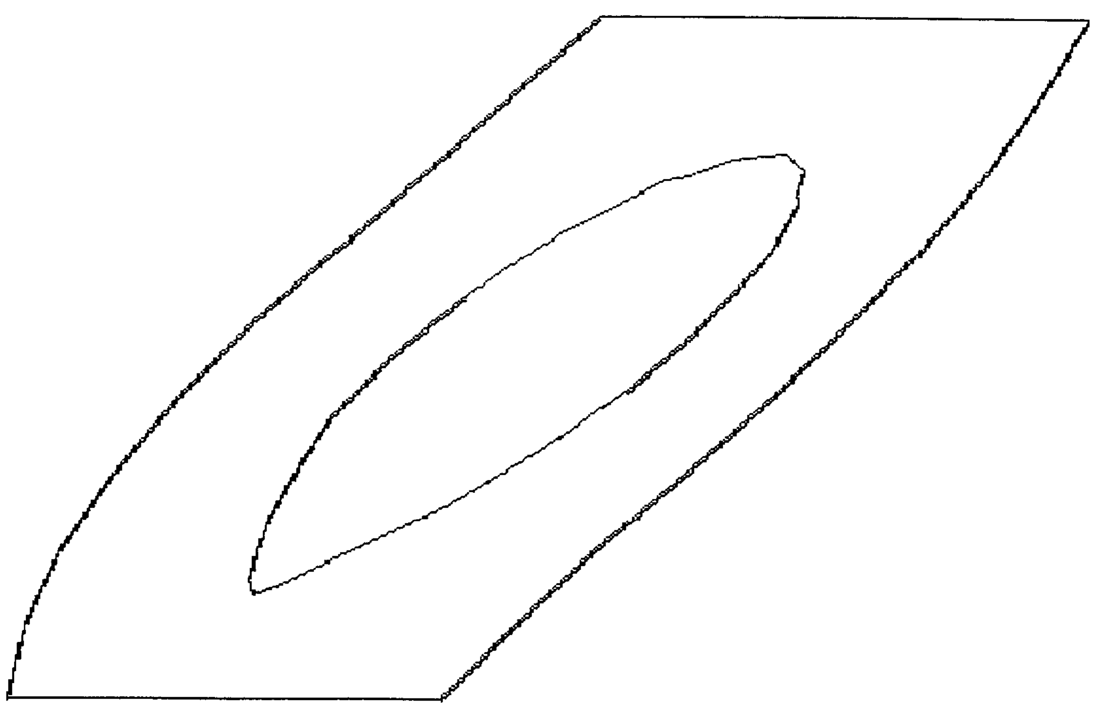


Figure B.28: Blade-to-Hub Root Intersection

Figure B.28: Blade-to-Hub Root Intersection

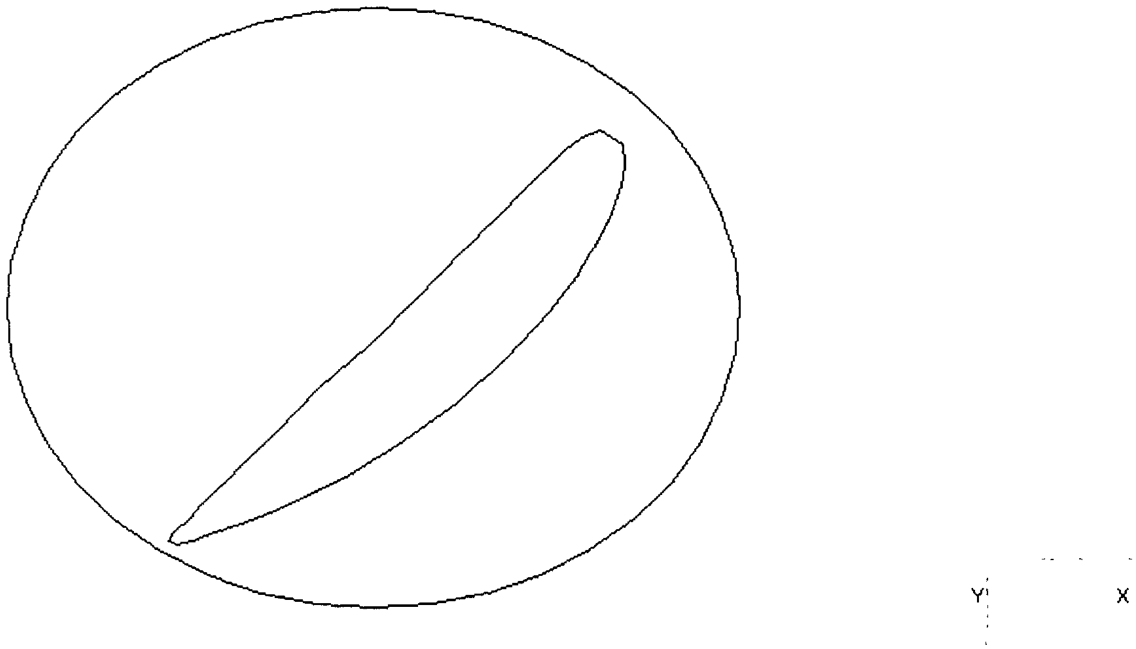


Figure B.29: Blade-to-Palm Root Intersection

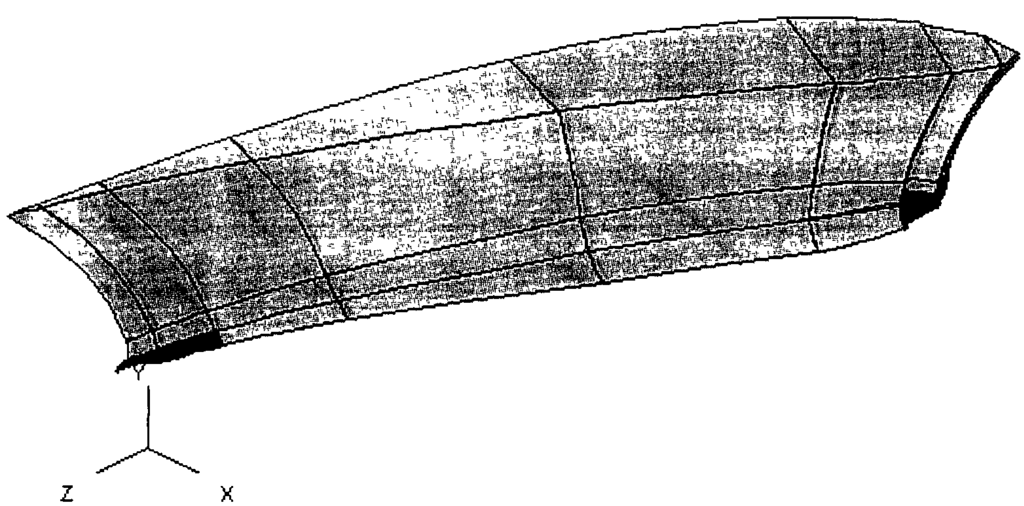


Figure B.30(a): Fillet

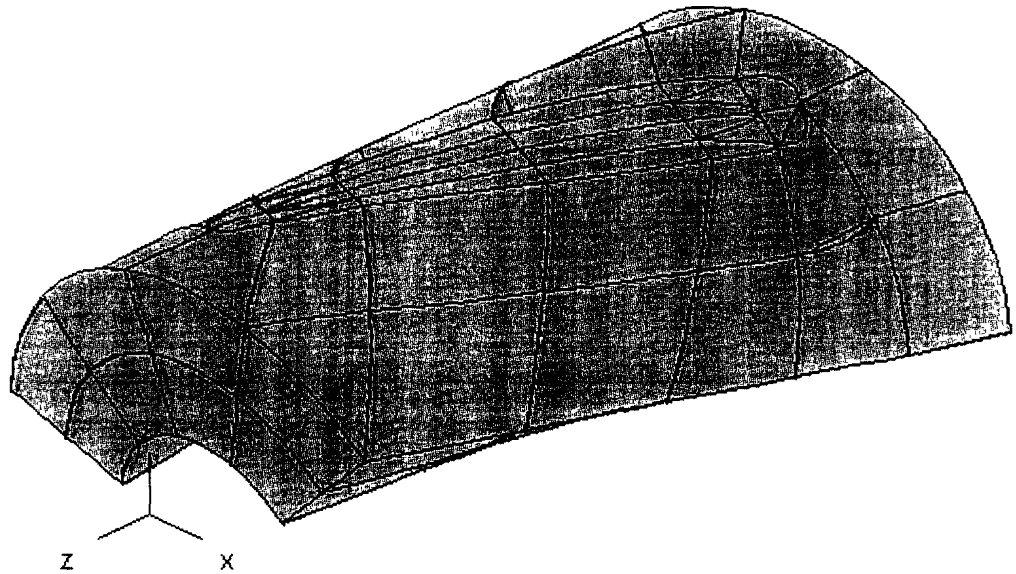


Figure B.30(b): Hub

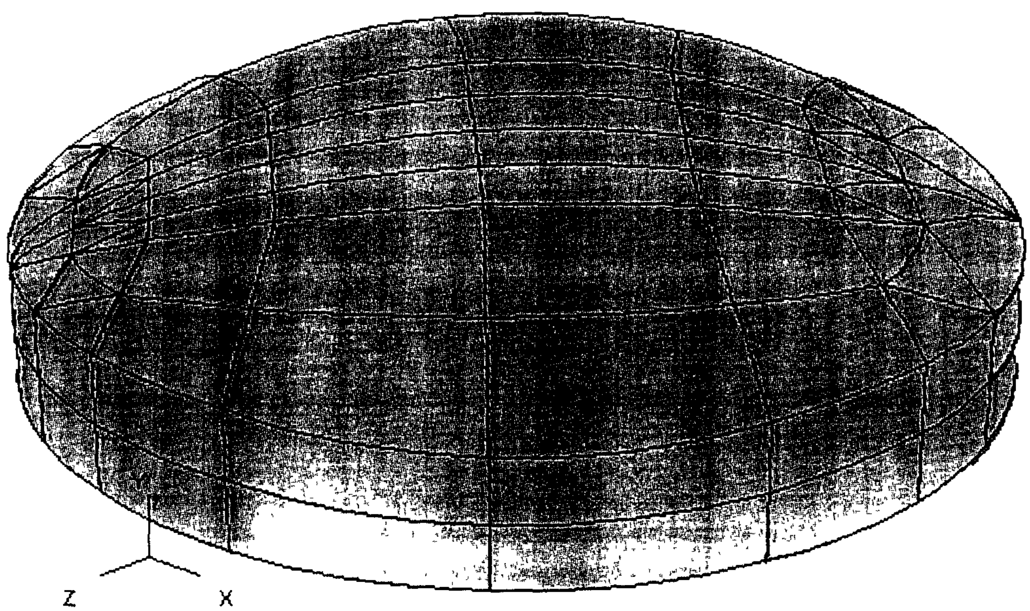


Figure B.30(c): Palm

Figure B.30: Finite Element Representation of Root Models: (a) Fillet; (b) hub; (c) Palm

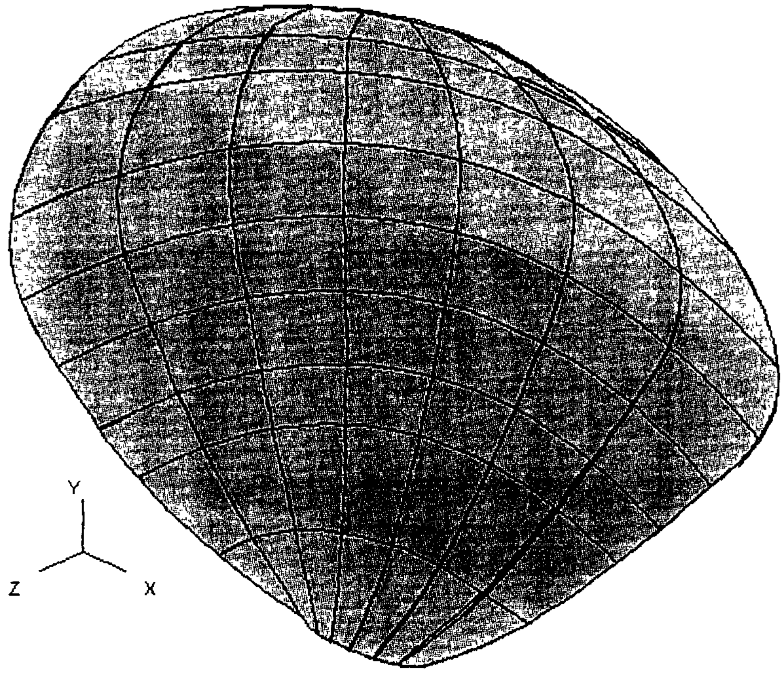


Figure B.31: Finite Element Representation of Blade Model

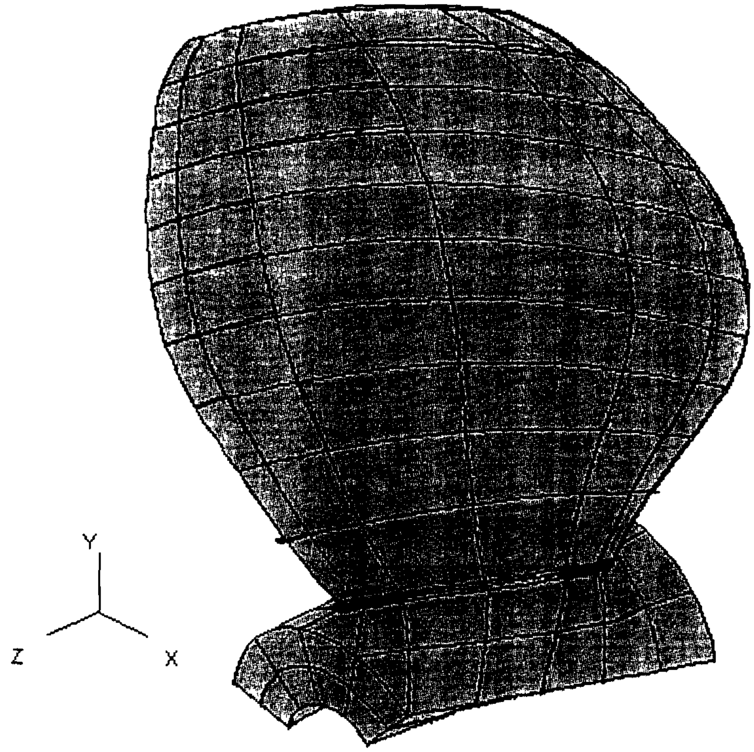


Figure B.32(a) Blade-Fillet-Hub

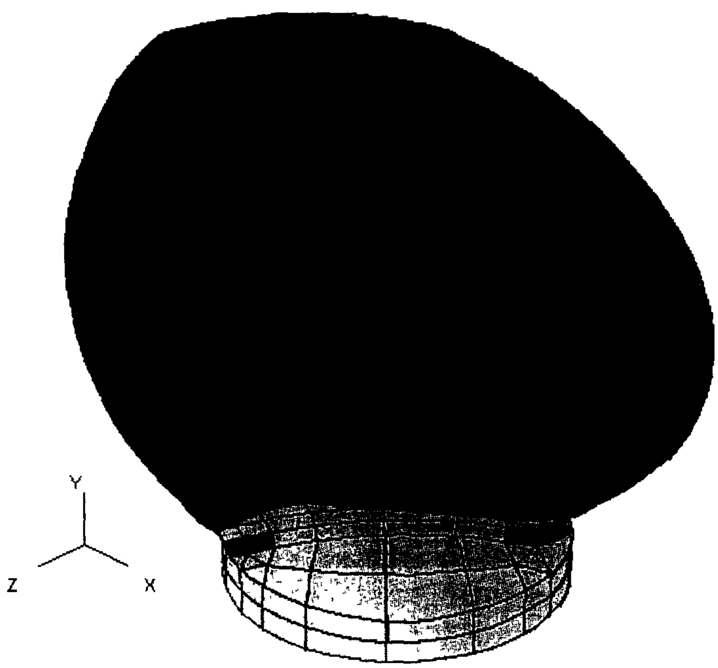
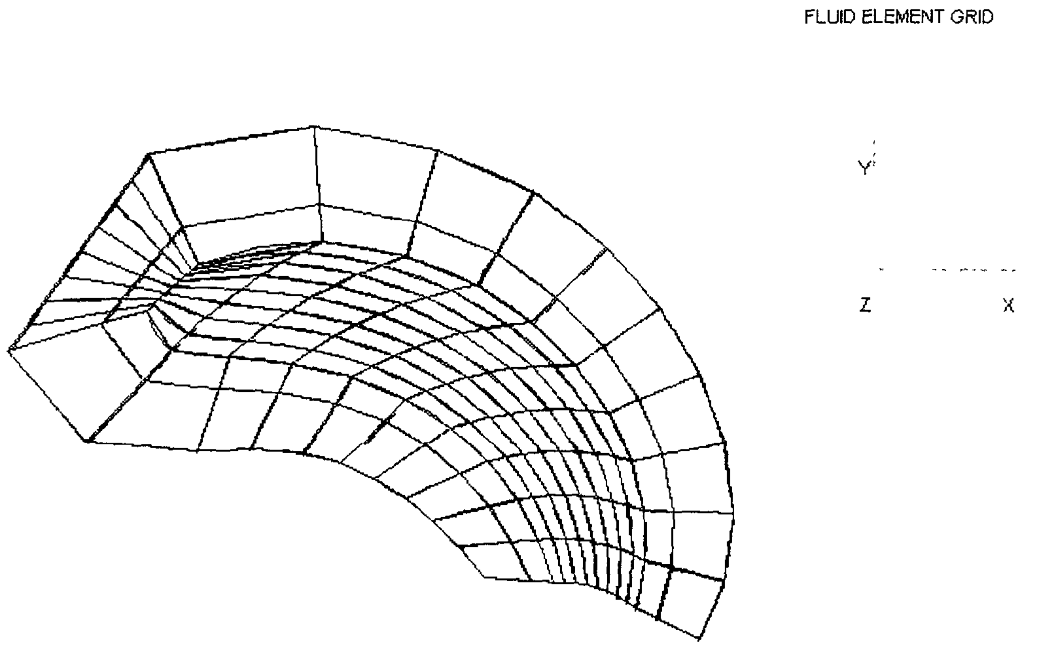
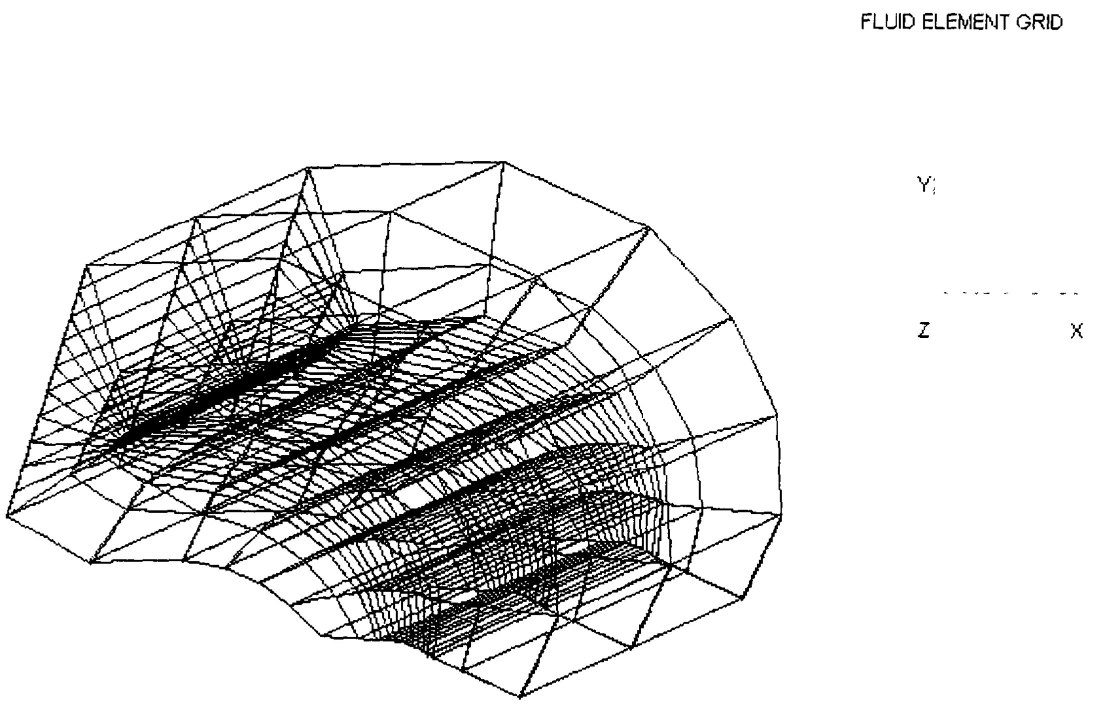


Figure B.32(b): Blade-Fillet-Palm

Figure B.32: FE Sub-structured Models: (a) Blade-Fillet-Hub; (b) Blade-Fillet-Palm



(a)



(b)

Figure B.33: Fluid FE Models (a) Thin Fluid Model, (b) Full 3D Fluid Model

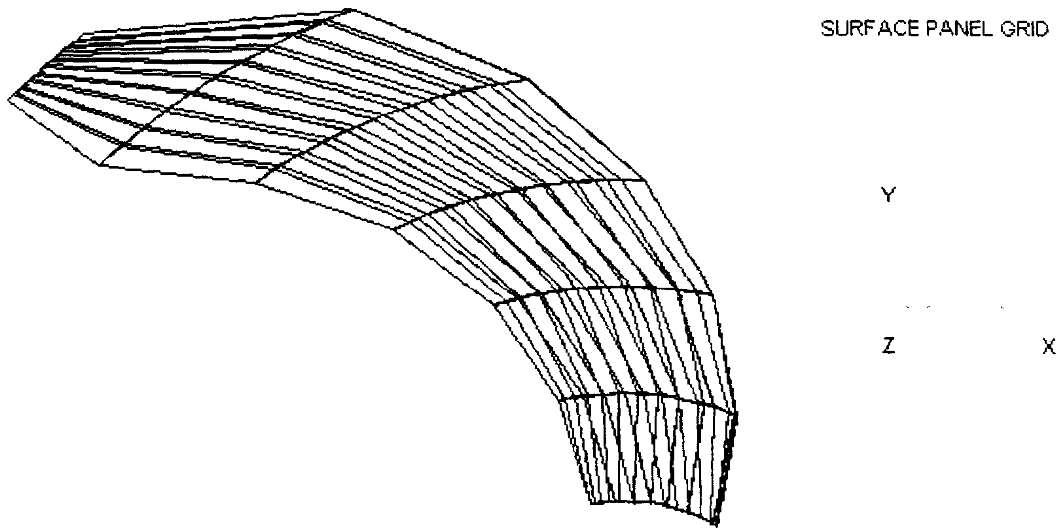


Figure B.33(c) Surface Panel

Figure B.33: Fluid Finite Element Models: (a) Thin Fluid; (b) Full 3-D Blade FE; (c) Surface Panel

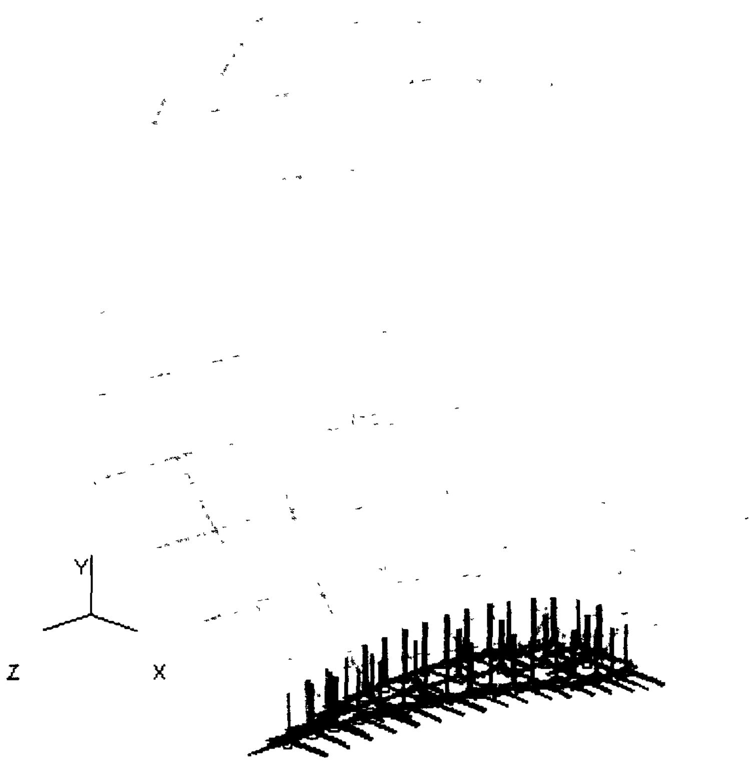


Figure B.34: Blade Finite Element Model Showing Boundary Conditions

APPENDIX C: SAMPLE PLOTS OF PVAST RESULTS

Figure C1: Radial Force Vectors in Typical Propeller Blade

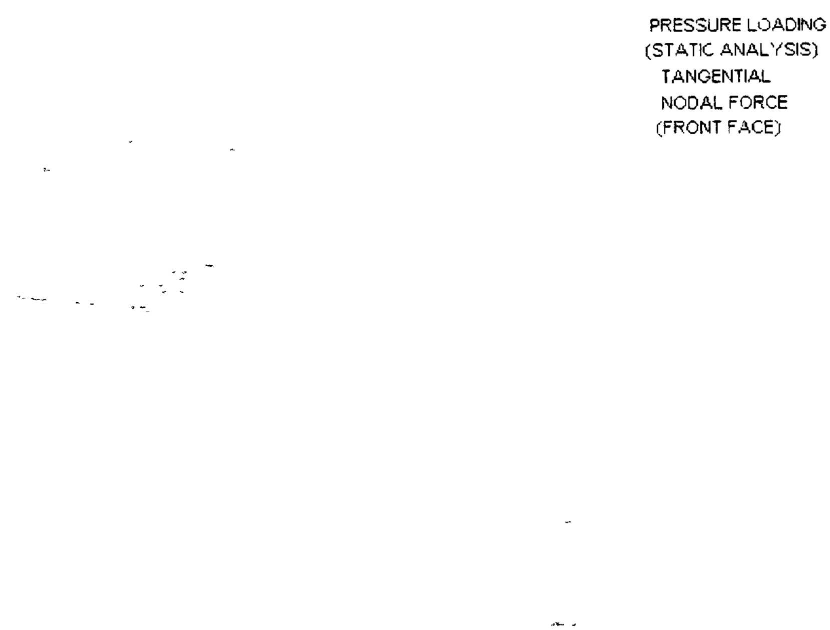


Figure C2: Tangential Force Vectors in Typical Propeller Blade

Figure C3: Normal Force Vectors in Typical Propeller Blade

Figure C4: Combined Force Vectors for Typical Propeller Blade

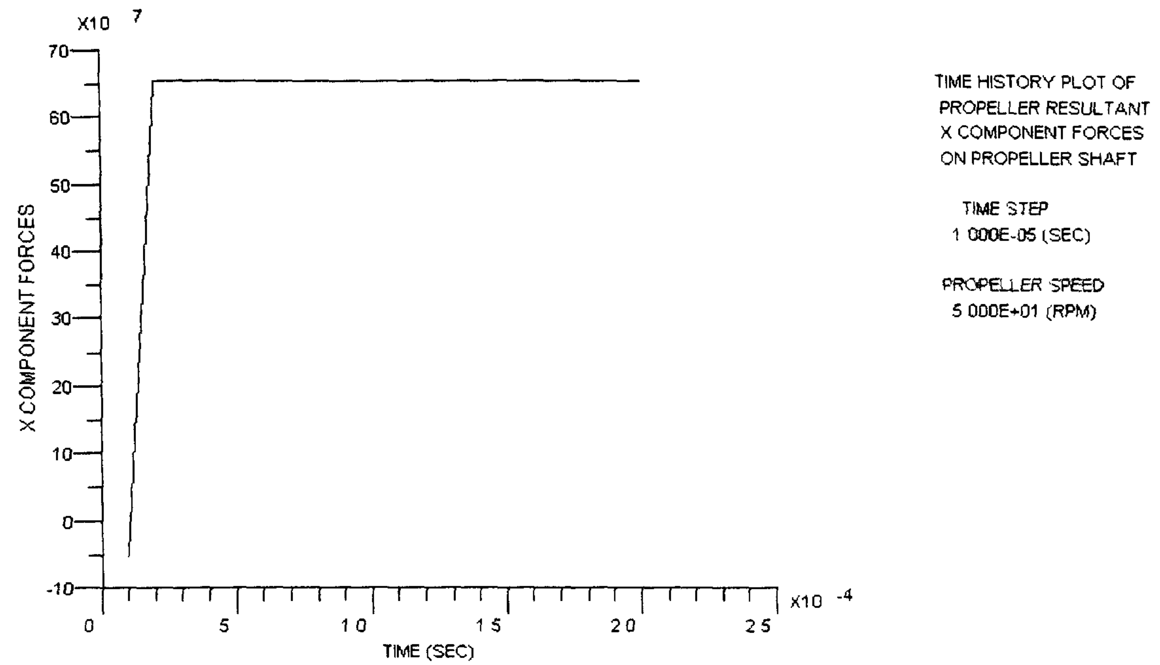


Figure C5: Resultant Force-Time History for Typical Propeller Blade

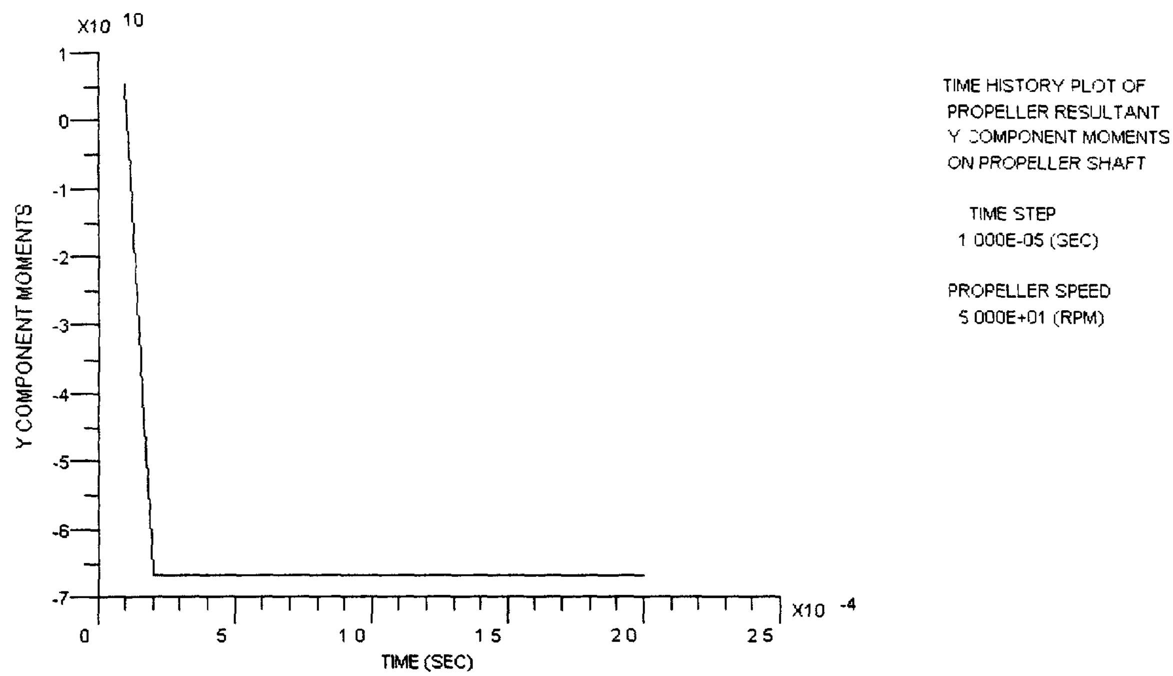


Figure C6: Resultant Moment-Time History for Typical Propeller Blade

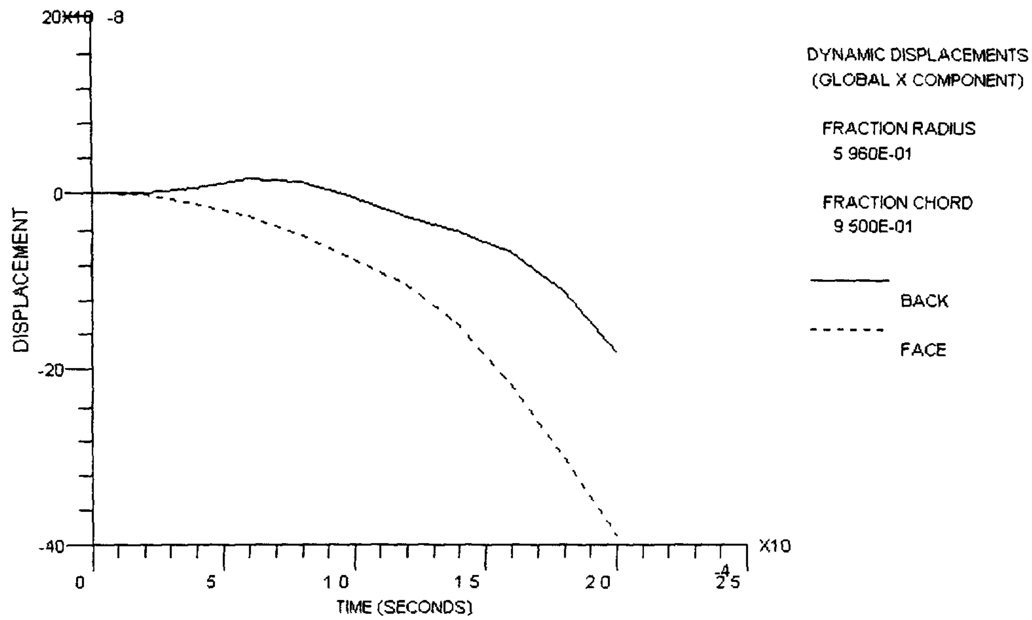


Figure C7: Displacement-Time History for Typical Propeller Blade

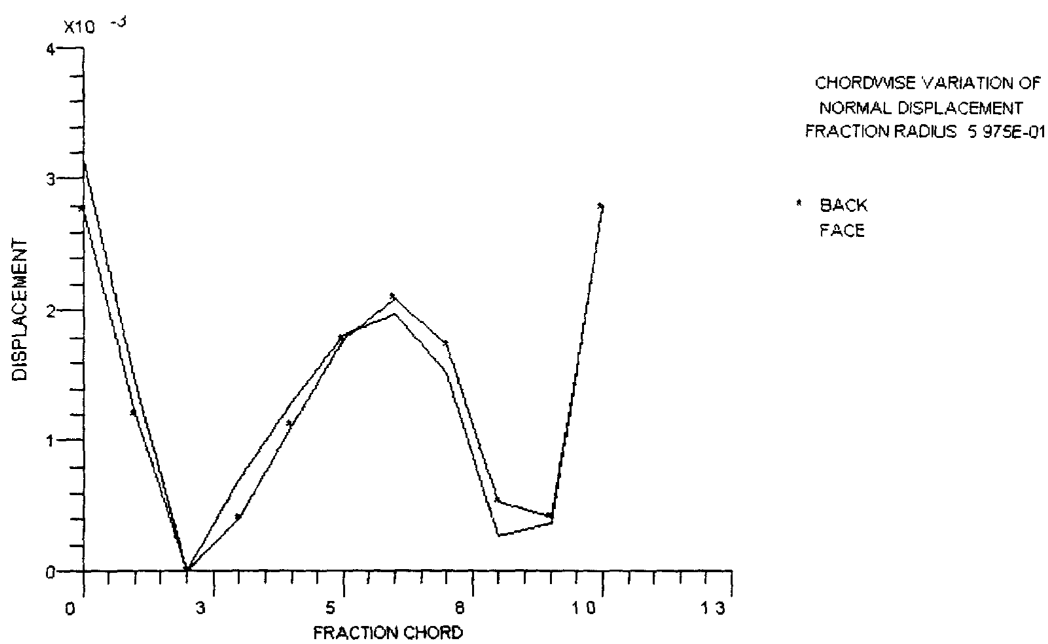


Figure C8: Chordwise Normal Displacement Plot for Typical Propeller Blade

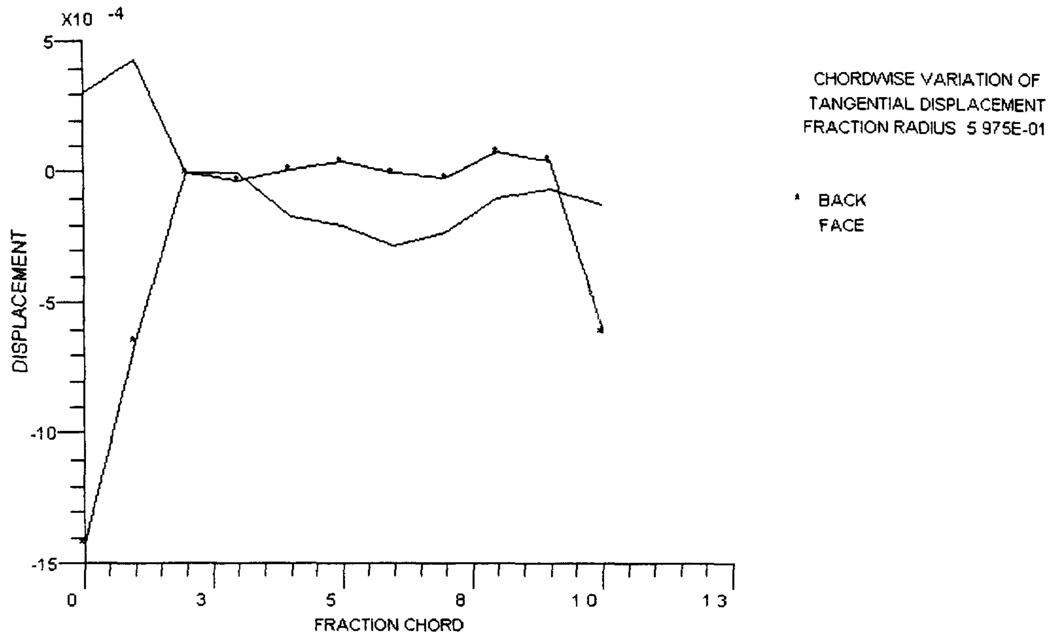


Figure C9: Chordwise Tangential Displacement Plot for Typical Propeller Blade

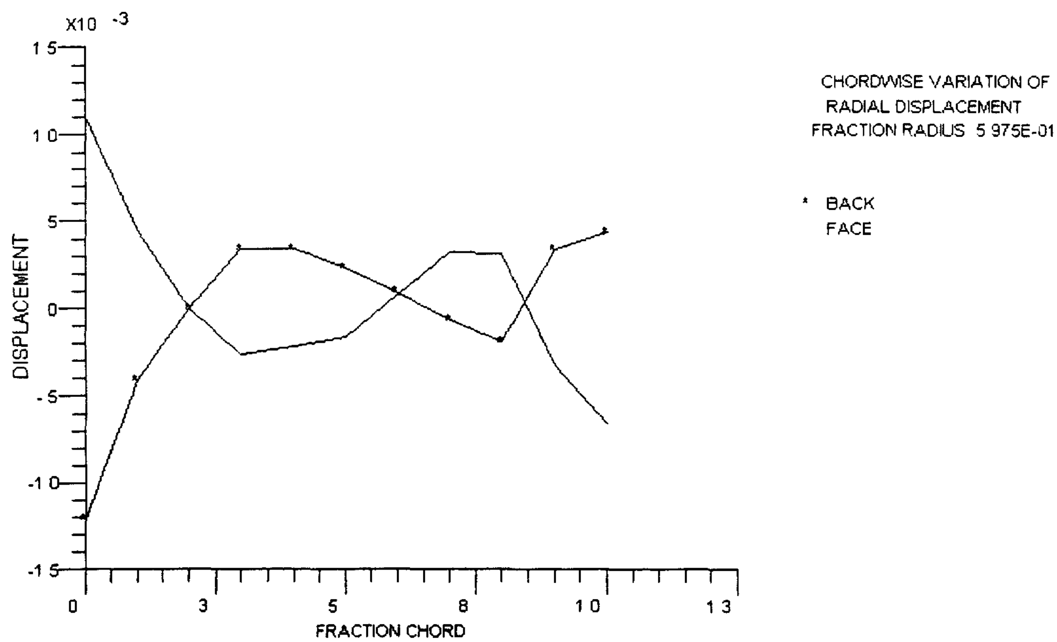


Figure C10: Chordwise Radial Displacement Plot for Typical Propeller Blade

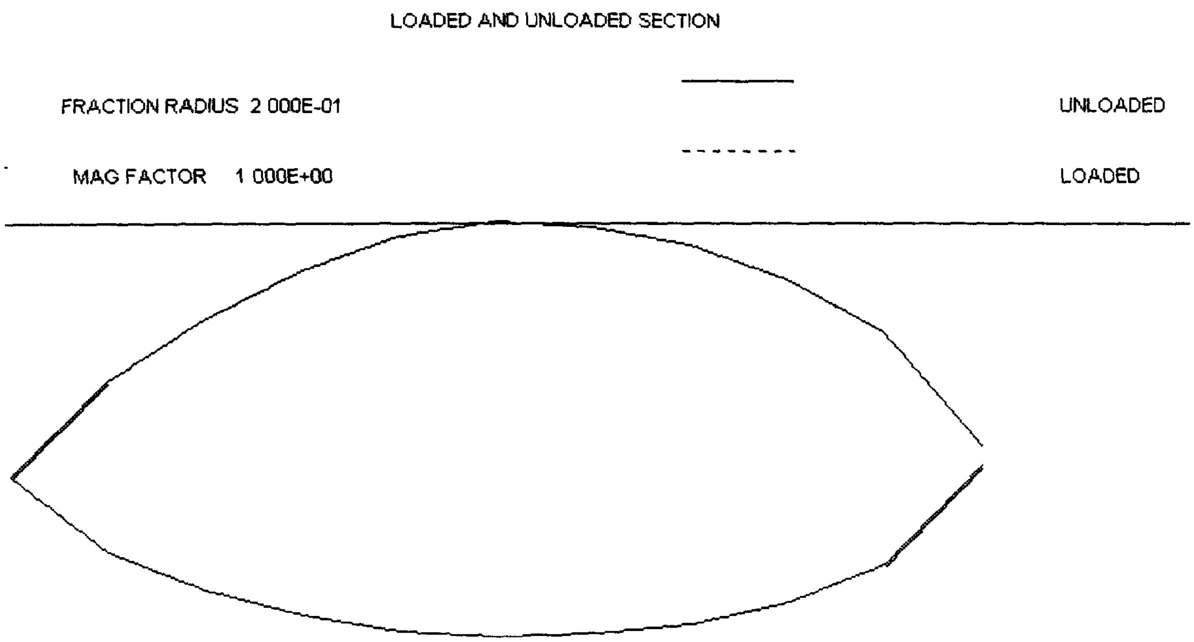


Figure C11: Blade Section Displacement for Typical Propeller Blade

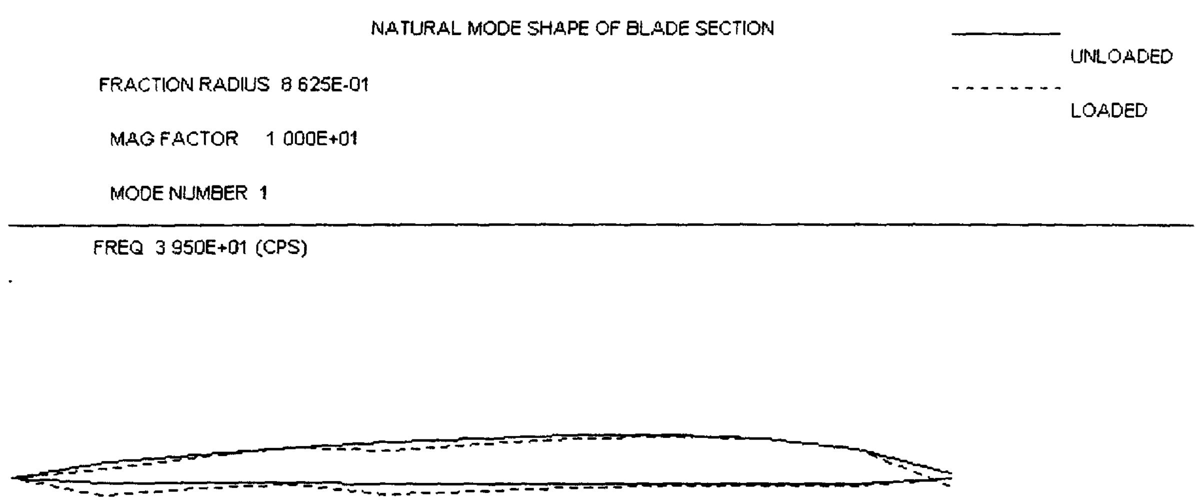


Figure C12: Blade Section Mode Shape for Typical Propeller Blade

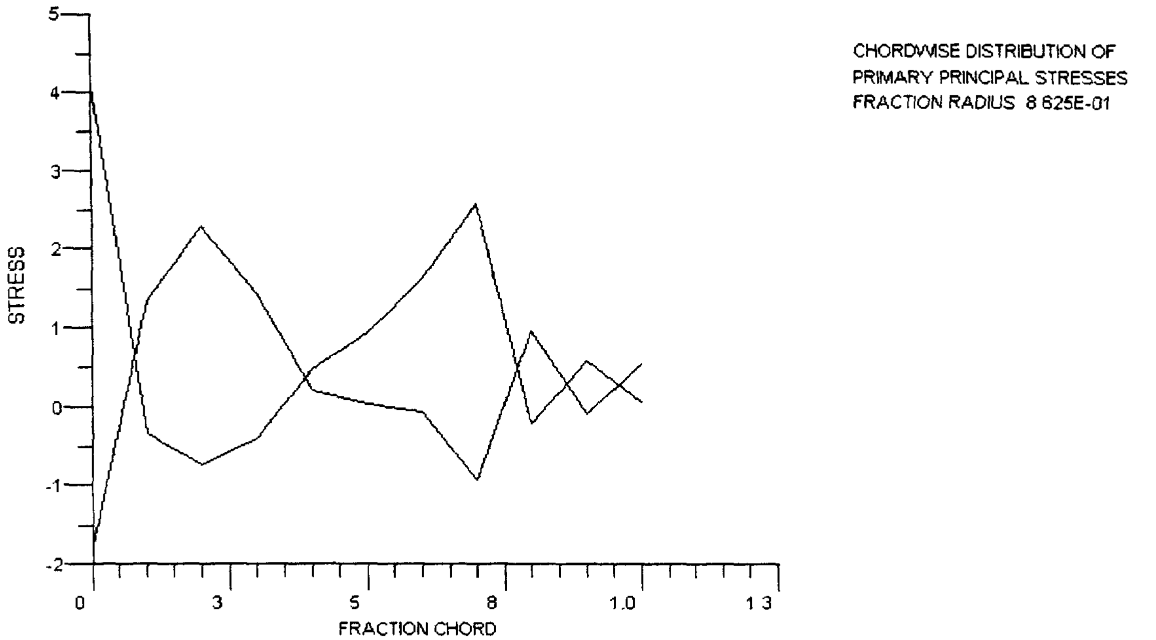


Figure C13: Chordwise Variation of Primary Principal Stresses for Typical Propeller Blade

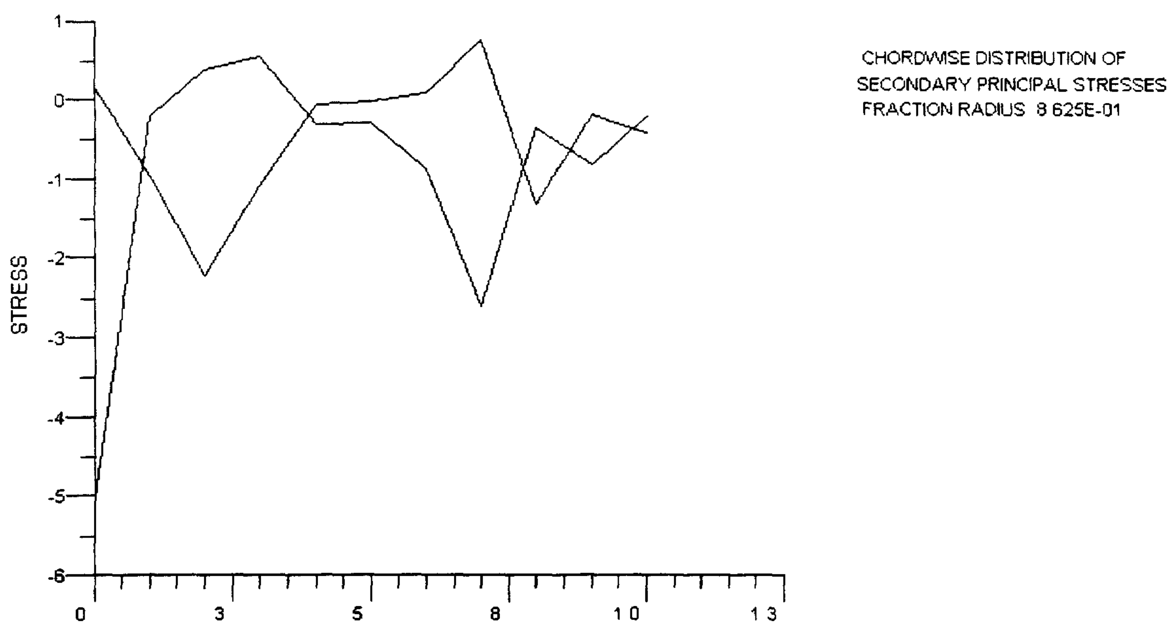


Figure C14: Chordwise Variation of Secondary Principal Stresses for Typical Propeller Blade

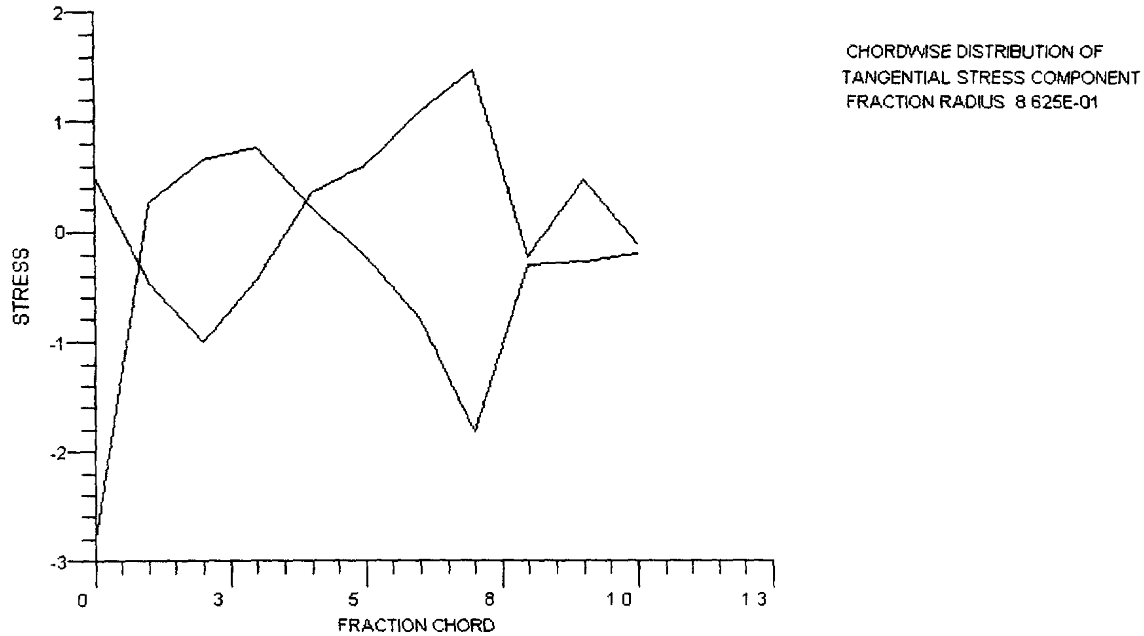


Figure C15: Chordwise Variation of Tangential Stresses for Typical Propeller Blade

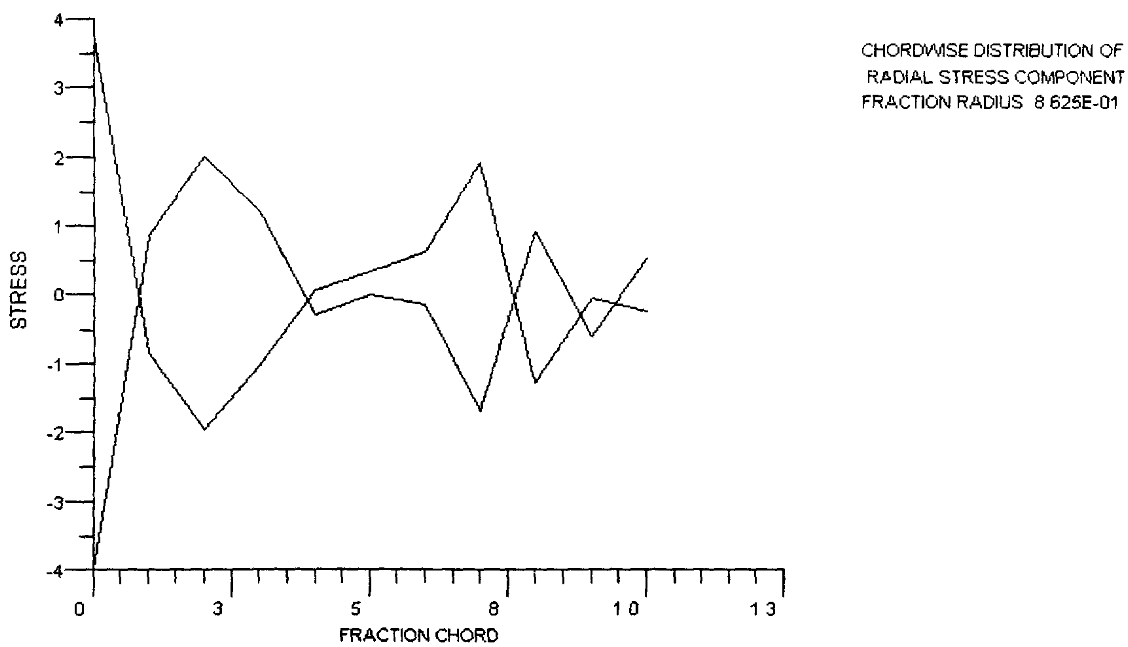


Figure C16: Chordwise Variation of Radial Stresses for Typical Propeller Blade

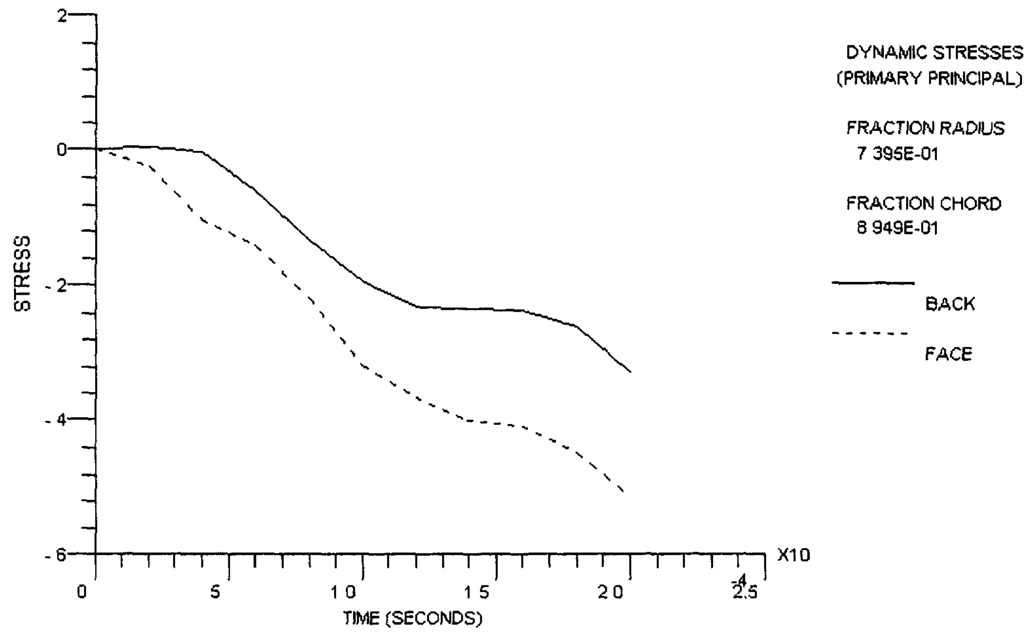


Figure C.17: Stress Time History of Typical Propeller Blade

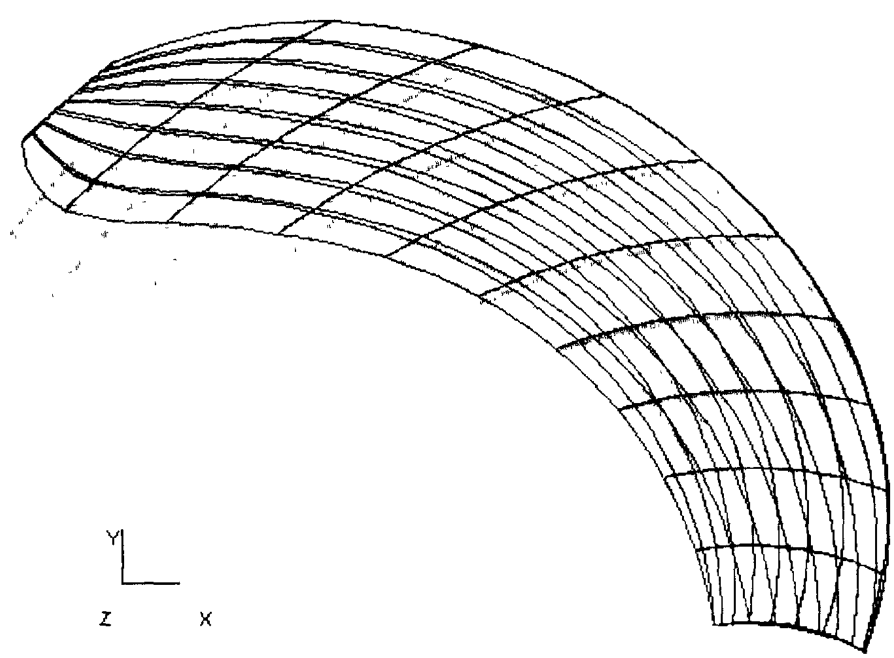


Figure C.18: Static Deformation of Blade Using General Plotting Option

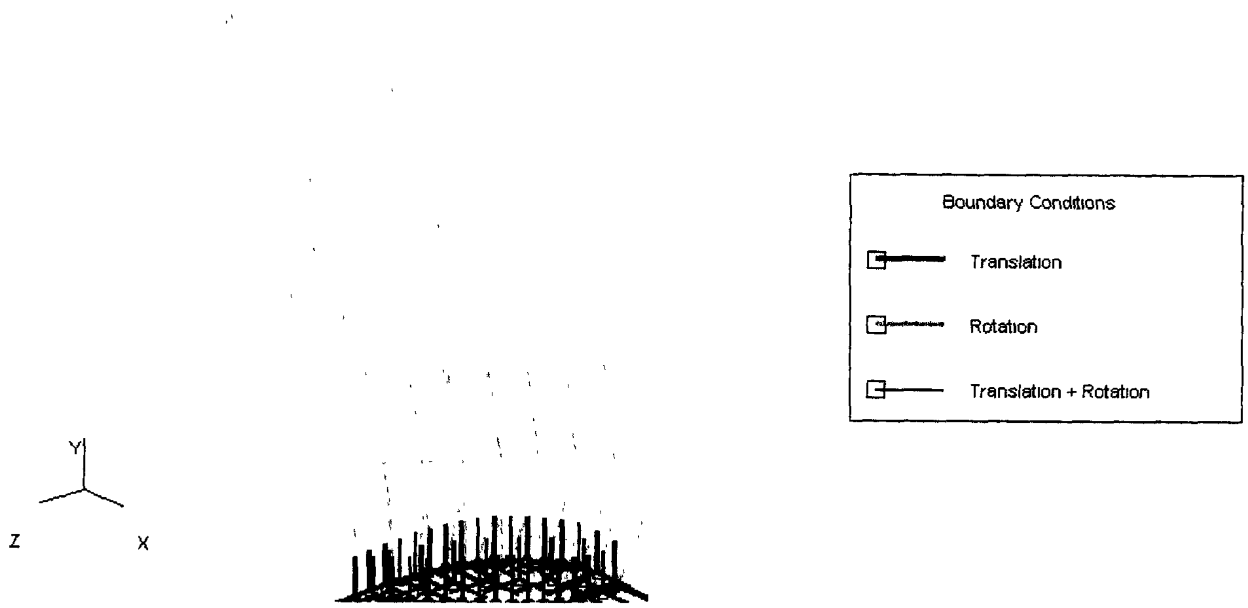


Figure C.19: Vibration Mode Shapes of Blade Using General Plotting Option

Figure C.20: Stress Contour Plot Using General Plotting Option

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM
(highest classification of Title, Abstract, Keywords)

DOCUMENT CONTROL DATA

(Security classification of title, body of abstract and indexing annotation must be entered when the overall document is classified)

1. ORIGINATOR (the name and address of the organization preparing the document.. Organizations for whom the document was prepared, e g. Establishment sponsoring a contractor's report, or tasking agency, are entered in section 8.)

MARTEC Limited
1888 Brunswick Street, Suite 400,
Halifax, Nova Scotia, Canada, B3J 3J8

2. SECURITY CLASSIFICATION (overall security classification of the document including special warning terms if applicable)

UNCLASSIFIED

3. TITLE (the complete document title as indicated on the title page Its classification should be indicated by the appropriate abbreviation (S,C,R or U) in parentheses after the title).

PVAST PROPELLER VIBRATION AND STRENGTH ANALYSIS PROGRAM VERSION 7.3 USER'S MANUAL

4 AUTHORS (Last name, first name, middle initial If military, show rank, e g. Doe, Maj. John E.)

Koko,T.S., Palmeter, M.F., Chernuka, M.W.

5 DATE OF PUBLICATION (month and year of publication of document)

March 2001

6a NO .OF PAGES (total containing information Include Annexes, Appendices, etc).

145

6b NO OF REFS (total cited in document)

8

7 DESCRIPTIVE NOTES (the category of the document, e g. technical report, technical note or memorandum. If appropriate, enter the type of report, e g. interim, progress, summary, annual or final. Give the inclusive dates when a specific reporting period is covered)

CONTRACTOR REPORT

8 SPONSORING ACTIVITY (the name of the department project office or laboratory sponsoring the research and development. Include address)

Defence Research Establishment Atlantic
PO Box 1012
Dartmouth, NS, Canada B2Y 3Z7

9a. PROJECT OR GRANT NO. (if appropriate, the applicable research and development project or grant number under which the document was written. Please specify whether project or grant).

Project 1gc11

9b. CONTRACT NO (if appropriate, the applicable number under which the document was written).

W7707-7-4689/001/HAL

10a ORIGINATOR'S DOCUMENT NUMBER (the official document number by which the document is identified by the originating activity. This number must be unique to this document.)

none

10b OTHER DOCUMENT NOs (Any other numbers which may be assigned this document either by the originator or by the sponsor.)

DREA CR 2000-152

11. DOCUMENT AVAILABILITY (any limitations on further dissemination of the document, other than those imposed by security classification)

(X) Unlimited distribution
(   ) Defence departments and defence contractors; further distribution only as approved
(   ) Defence departments and Canadian defence contractors; further distribution only as approved
(   ) Government departments and agencies; further distribution only as approved
(   ) Defence departments; further distribution only as approved
(   ) Other (please specify):

12. DOCUMENT ANNOUNCEMENT (any limitation to the bibliographic announcement of this document This will normally correspond to the Document Availability (11) However, where further distribution (beyond the audience specified in (11) is possible, a wider announcement audience may be selected).

Unlimited

UNCLASSIFIED

13. ABSTRACT (a brief and factual summary of the document It may also appear elsewhere in the body of the document itself It is highly desirable that the abstract of classified documents be unclassified Each paragraph of the abstract shall begin with an indication of the security classification of the information in the paragraph (unless the document itself is unclassified) represented as (S), (C), (R), or (U) It is not necessary to include here abstracts in both official languages unless the text is bilingual)

This report is a user's manual for the finite element analysis code called PVAST (Propeller Vibration And STrength) that is used for prediction of stress and vibration in marine propellers. PVAST can automatically generate propeller finite element models from basic propeller geometry defined by the orientation of 2D "wrapped" blade sections at specified radii. A number of different models can be considered including; a single blade, a single blade with fillet, a blade-fillet-palm model, a blade-fillet-hub segment model, and multiple blade and hub model. The types of structural finite element analyses that can be conducted include; static analysis with user-defined blade pressure distributions and point loads, natural frequency analysis in air and in water, time domain analysis with applied loads and support motion, response spectrum analysis and frequency response analysis. The program provides 2D and 3D plotting of blade geometry, finite element models and post-processing to provide visualization of predicted finite element model displacements, stresses and mode shapes. The code has been modified recently to include a "Windows" graphical user interface implemented with a combination of GKS and MS Visual C++. This provides a more "friendly" user interface and is a major change in the look and feel of PVAST Version 7.3 compared with earlier versions of the code.

14. KEYWORDS, DESCRIPTORS or IDENTIFIERS (technically meaningful terms or short phrases that characterize a document and could be helpful in cataloguing the document They should be selected so that no security classification is required Identifiers, such as equipment model designation, trade name, military project code name, geographic location may also be included If possible keywords should be selected from a published thesaurus e g Thesaurus of Engineering and Scientific Terms (TEST) and that thesaurus-identified If it not possible to select indexing terms which are Unclassified, the classification of each should be indicated as with the title)

marine propeller
finite element
stress
vibration

Defence R&D Canada
is the national authority for providing
Science and Technology (S&T) leadership
in the advancement and maintenance
of Canada's defence capabilities.

R et D pour la défense Canada
est responsable, au niveau national, pour
les sciences et la technologie (S et T)
au service de l'avancement et du maintien des
capacités de défense du Canada.

DEFENCE R&D DÉFENSE

www.drdc-rddc.dnd.ca