Image Cover Sheet

CLASSIFICATION	SYSTEM NUMBER 516541
UNCLASSIFIED	

TITLE

PVAST propeller vibration and strength analysis program version 7.3 examples manual

System Number: 516541

Patron Number:

Requester:

Notes:

DSIS Use Only:

Deliver to:

This page is left blank

This page is left blank

REPRODUCTION QUALITY NOTICE

Copy No. 11

DEFENCE R&D DÉFENSE

PVAST PROPELLER VIBRATION AND STRENGTH ANALYSIS PROGRAM VERSION 7.3 EXAMPLES MANUAL

Koto, 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 2000-153
March 2001

National Defence Défense nationale

Canada

Copy No: ________

PVAST Propeller Vibration and Strength Analysis Program Version 7.3 Examples 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 2000-153
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 an examples 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. This manual provides a number of example problems, considered with PVAST Version 7.3 for a range of model types, analysis options and results.

Résumé

Le présent rapport est un manuel d'exemples 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. Le manuel donne un certain nombre d'exemples de problèmes, basés sur la version 7.3 du PVAST et qu'on prévoit pour une série de types modèles, d'options d'analyses et de résultats.

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 Examples Manual. DREA CR 2000-153 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 2000-153 Centre pour la Recherche de la D'efence Atlantique.

TABLE OF CONTENTS

TABLE OF CONTENTS con't

TABLE OF CONTENTS con't

TABLE OF CONTENTS con't

LIST OF TABLES

LIST OF FIGURES

1. INTRODUCTION

1.1 Background

This document provides example problems to demonstrate the use of program PVAST version 7.3. 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 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.

1.2.1 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.

1.2.2 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

1.2.3 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 models • 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

1.2.4 Operating System

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

1.3 Applicable Documents

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

- T.S. Koko, M.F. Palmeter, M.W. Chernuka, and T. MacFarlane. "PVAST. Propeller Vibration and Stress Analysis by Finite Element Methods Version 7.3 User's Manual." Martec Limited, June 2000.

- 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.

1.4 Organization of This Manual

The remainder of this manual is organized as follows:

Chapter 2: Provides examples on how to generate the geometry of propeller blades. The DREA propeller blades named TRUMP, NRC45, P5363 and P4388 are used for the demonstrations.

Chapter 3: Demonstrates how to generate *Blade Only* finite element models, perform natural frequency analysis in air and to display the results, using the NRC45 propeller blade. Finite element models (*Blade Only*) of P5363 and P4388 propeller blades are also presented.

Chapter 4: Demonstrates how to generate fluid finite element models, perform natural frequency analyses in water and to display the results, using the NRC45 propeller blade. The development of fluid finite elements for the P5363 and P4388 propeller blades is also presented.

Chapter 5: Shows how to perform stress analysis of *Blade Only* finite element models and to display the results, using the P5363 propeller blade.

Chapter 6: Illustrates how to generate *Blade-Fillet-Hub* models, using the P5363, P4388 and NRC45 propeller blades

Chapter 7: Demonstrates how to generate *Blade-Fillet-Palm* models using another DREA propeller blade named PD280.

Chapter 8: Illustrates how to generate a two-dimensional hub model with a *Keyway*.

Chapter 9: Demonstrates how to generate air-channel models of the blade and perform stress analysis. The P5363 propeller blade is used for this purpose.

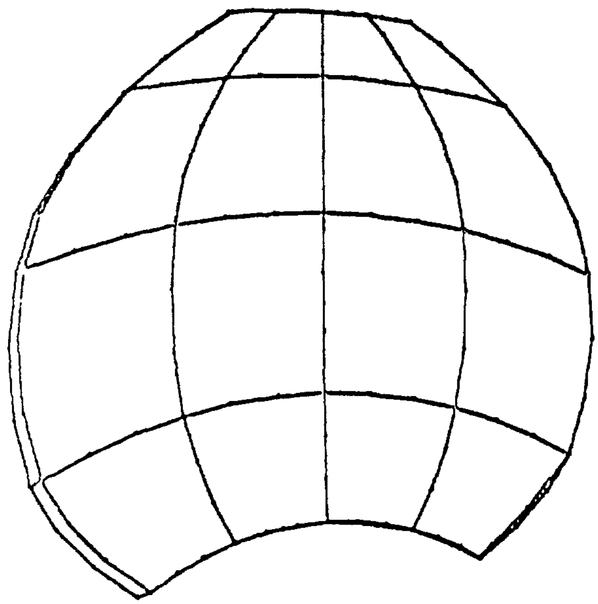


Figure 1.1: Blade Only Model

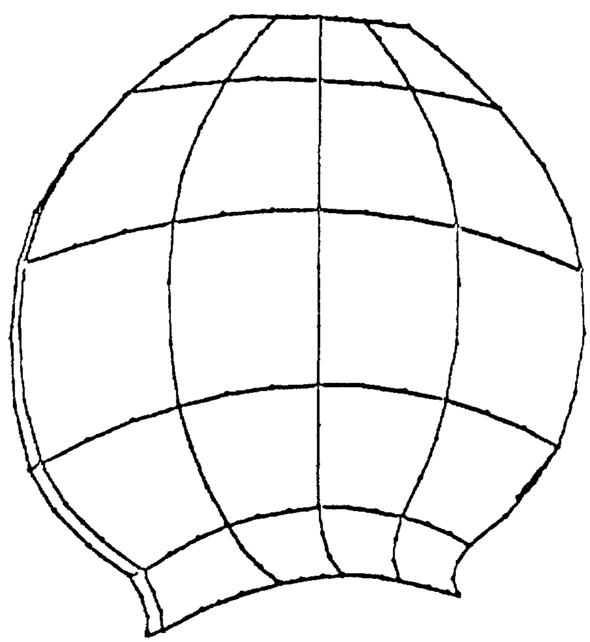


Figure 1.2: Blade-Fillet Model

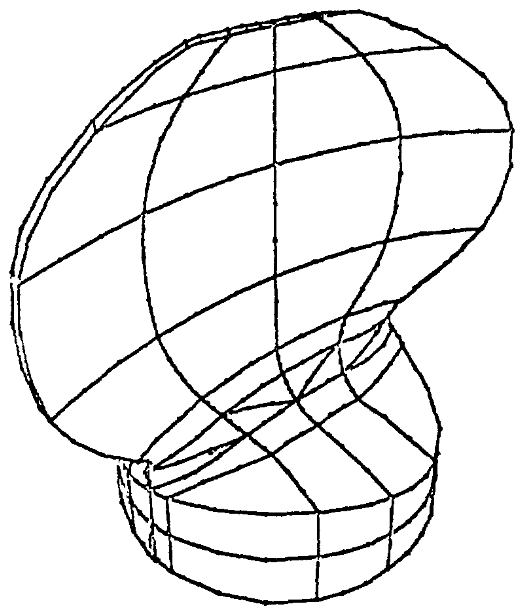


Figure 1.3: Blade-Fillet-Palm (Bolts as Rods) Model (shown without bolts)

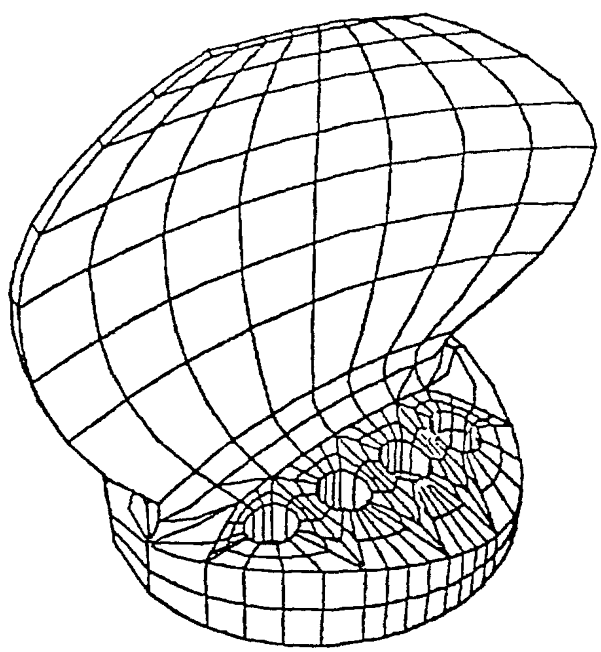


Figure 1.4: Blade-Fillet-Palm (Bolts as Superelements) Model

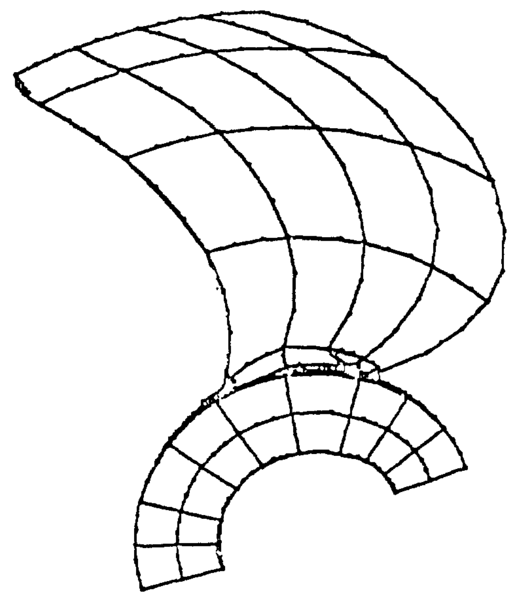


Figure 1.5: Blade-Fillet-Hub Model

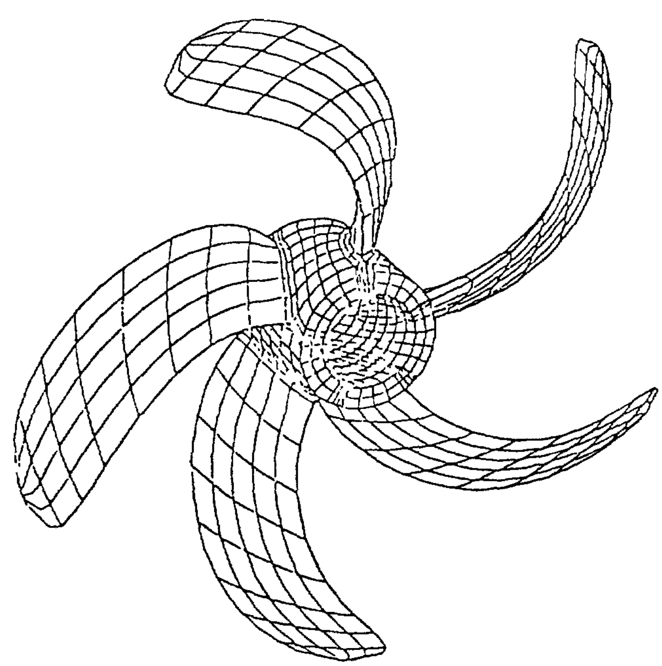


Figure 1.6: Multi-Blade Hub Model

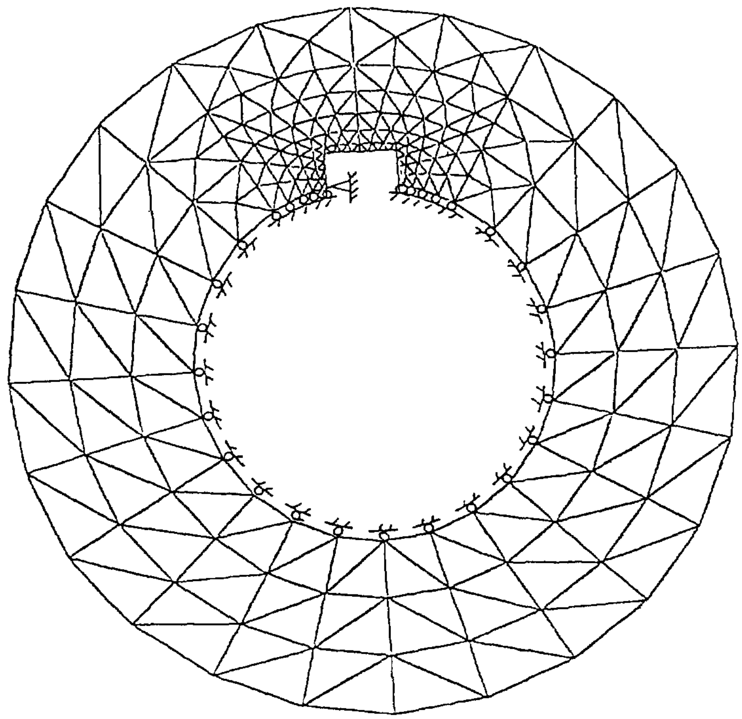


Figure 1.7: Keyway Model

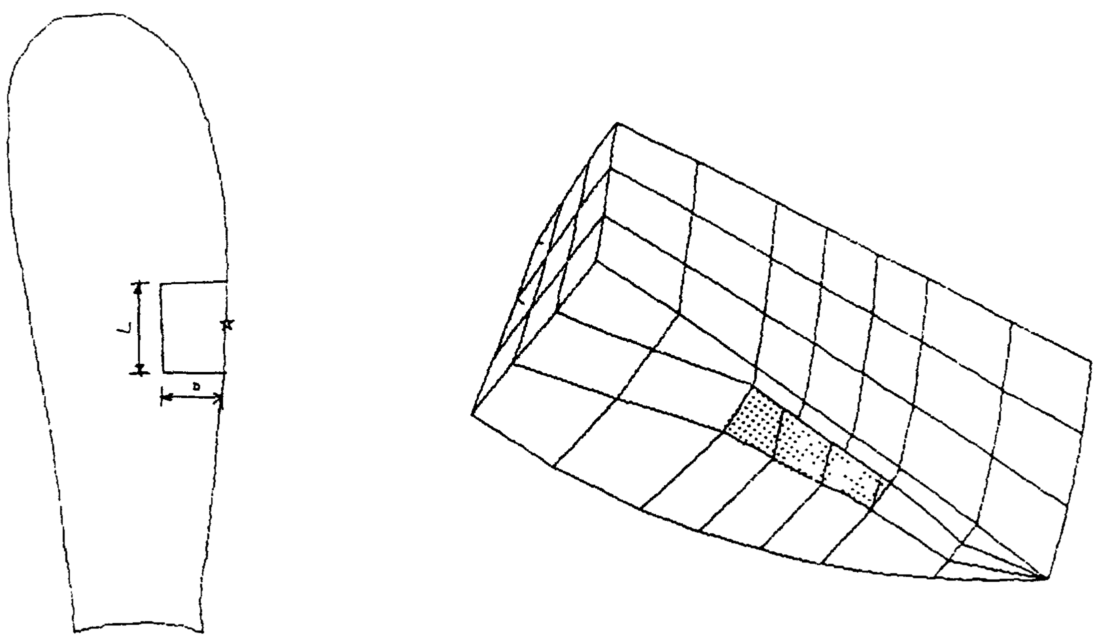


Figure 1.8: Air Channel Model

2. BLADE GEOMETRY GENERATION

2.1 Introduction

This chapter illustrates how to generate the geometry of propeller blades. This is considered to be the first main step of the blade finite element modelling process. This could be a fresh start or the blade geometry definition data may be available on a file called PREFIX.DAT, where PREFIX is the alphanumeric prefix name used to identify files for the current session. When the latter is the case, the data is presented in the PREFIX.DAT according to the format specified in Appendix A, and the user could proceed to edit/modify or graphically view the data as illustrated in this chapter. For a fresh start, the procedure illustrated in this chapter has to be followed.

Figure 2.1 shows the **Blade Geometry** Submenu. The steps involved are:

- Definition of Basic Data;
- Definition of Radial Data;
- Definition of Chordwise Data;
- Definition of Edge Data, if required; and
- Definition of Hub/Palm data, if required.

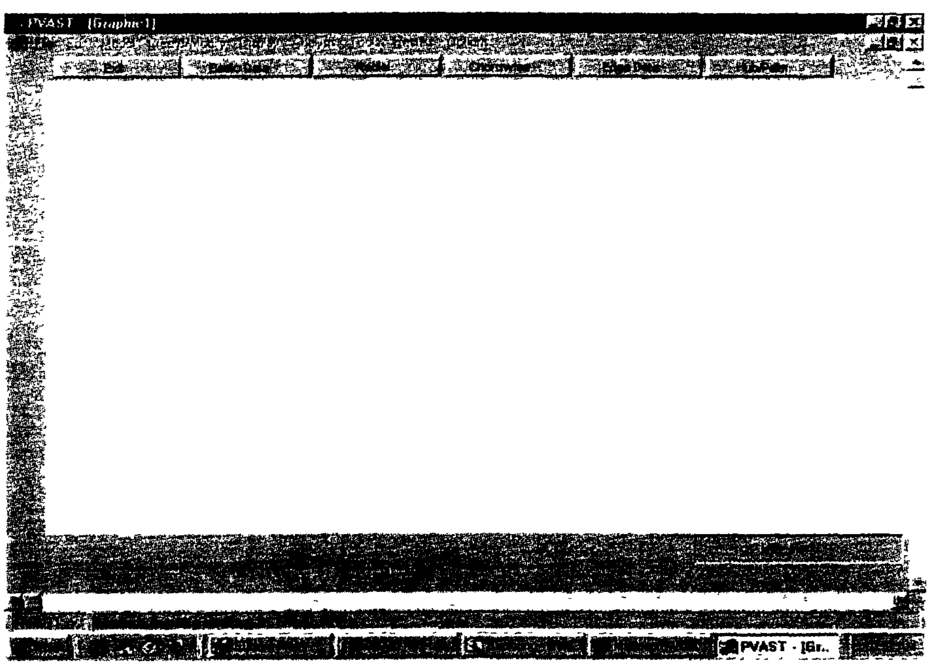


Figure 2.1: Blade Geometry Menu

It is preferable to go through the steps in the order shown. The blade data is defined through several edit tables, and graphical plots of the data and the blade geometric configuration are provided to aid the modelling process. In Section 2.2, details of the edit tables and the significant plots for DREA's TRUMP propeller blade are provided. Then in Sections 2.3 to 2.5, the data entered for generating three other blades, namely DREA's NRC45, P5363 and P4388 propeller blades are presented.

It should be noted that during the blade geometry definition process, the program creates several files that are required for plotting the data or required by other modules of PVAST. One of these files is the PREFIX.T16 file which is required for creating the finite element grid proportions [1]. This file may also serve as a starting point in the PVAST modelling process. When this is the case, the blade geometry definition step is not required, and the user should go directly to the FE grid proportions definition step. The format of the PREFIX.T16 file is also shown in Appendix A.

2.2 Geometry of TRUMP Propeller Blade

2.2.1 Basic Data

Figure 2.2 shows the basic propeller data entered for this blade. The form of data is offsets.

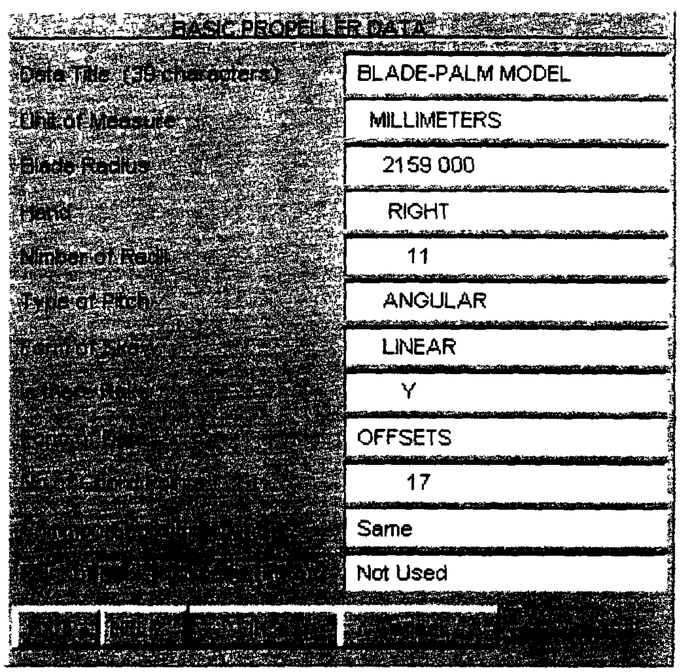


Figure 2.2: Basic Data for TRUMP Propeller Blade

2.2.2 Radial Data

Figure 2.3 shows the radial data entered for this blade.

RADIUS	CHORD	PITCH	SKEW	RAKE
2980	775.0810	48 0790	49 8000	5000
3000	779 8310	48 3320	45 8000	3.8000
3500	923 1180	50 0770	-48 9000	78 1000
4000	1059 2050	49 3200	-126 9000	139 9000
5000	1341 1710	45 1310	-196 6000	188 6000
6000	1611 9090	39 2260	-158 3000	155.3000
7000	1763 9030	34 2260	-49.2000	88 1000
8000	1734 1090	29 0150	118 5000	8 6000
9000	1427 9630	22 9970	324 7000	-55 2000
9500	1094 1810	19 4000	431 4000	-68 6000
9750	823 8740	17 3680	485 5000	-68 9000

OK Cancel HELP

Figure 2.3: Radial Data for TRUMP Propeller Blade

Figures 2.4 to 2.7 show the variations of the chord length, pitch, skew and rake along the radial direction.

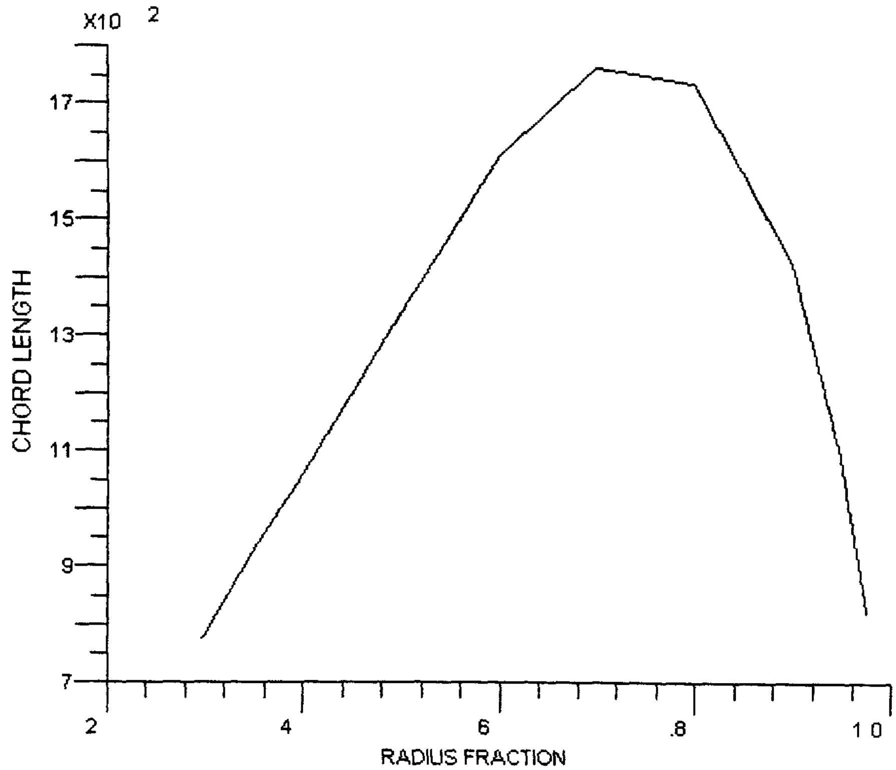


Figure 2.4 Plot of Chord Length vs Radial Fraction for TRUMP Propeller Blade

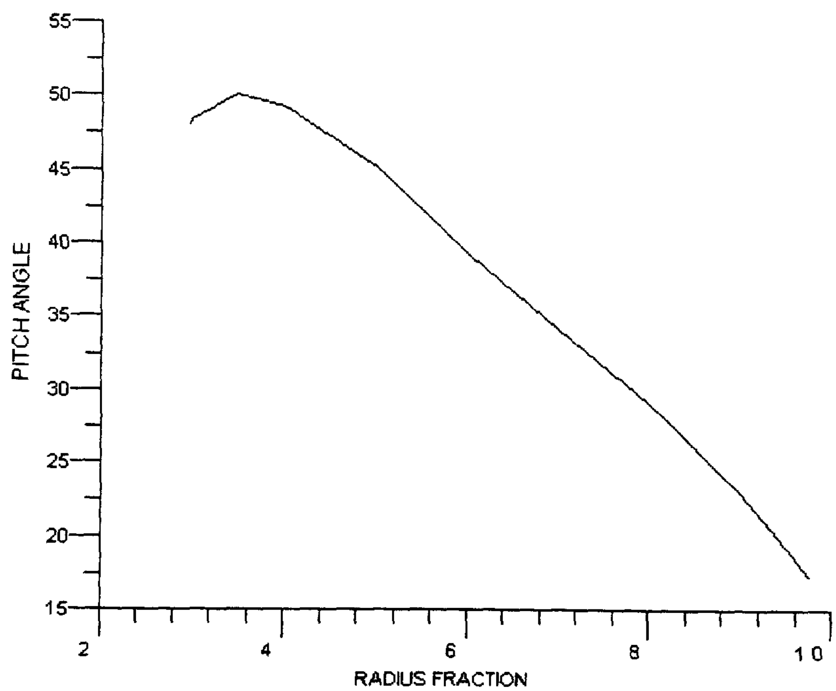


Figure 2.5: Plot of Pitch Angle vs Radial Fraction for TRUMP Propeller Blade

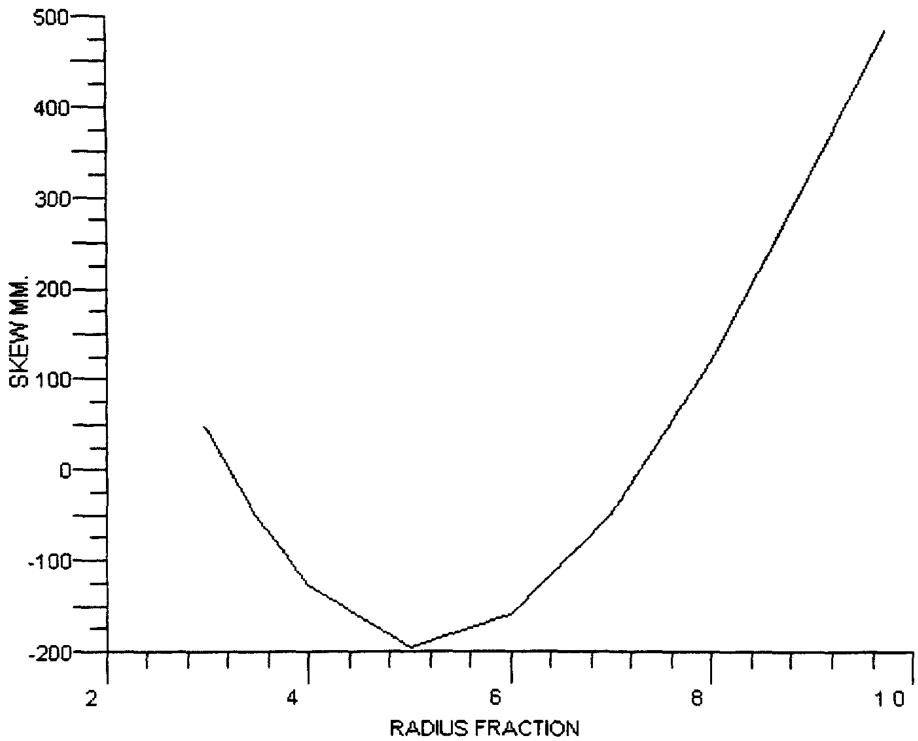


Figure 2.6: Plot of Skew vs Radial Fraction for TRUMP Propeller Blade

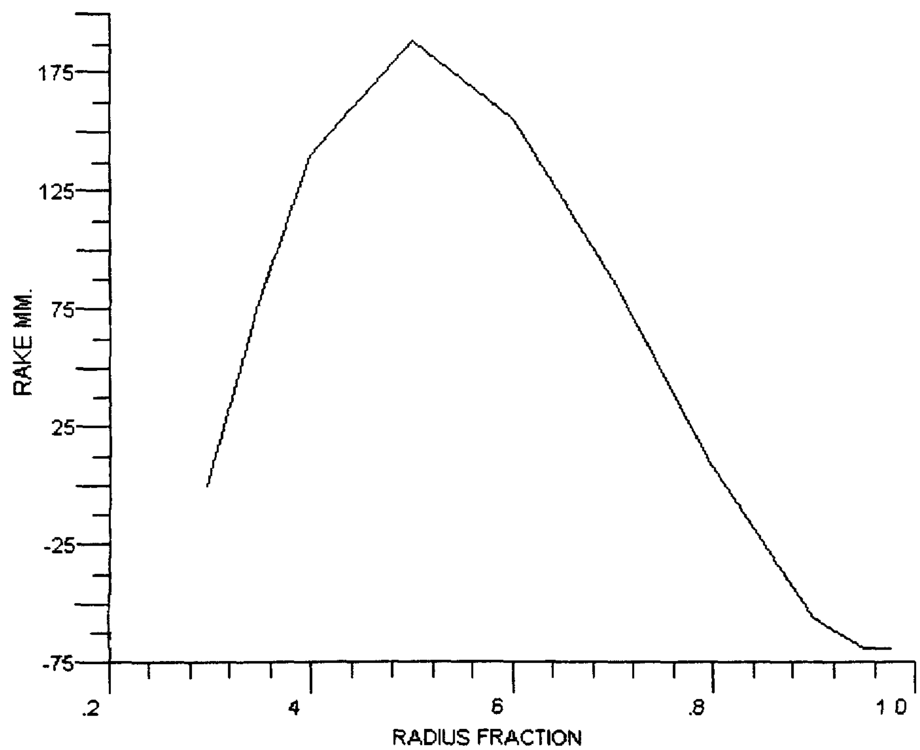


Figure 2.7: Plot of Rake vs Radial Fraction for TRUMP Propeller Blade

2.2.3 Cross Section Data

Figures 2.8 (a) to (k) show the cross section data at the 11 radial positions entered for this blade.

CROSS SECTION AT RADIUS [illegible]

CHORD FRACTION	BACK OFFSET	FACE OFFSET	CHORD FRACTION	BACK OFFSET	FACE OFFSET
0000	0000	0000	6000	-80 1000	85 9640
0100	-15 9000	17 5010	7000	-69 6000	74 4780
0250	-25 0000	27 3530	8000	-53 7000	57 3720
0500	-35 3000	38 4600	9000	-32 3000	34 7390
1000	-49 8000	54 0000	9500	-19 6000	21 2370
2000	-68 5000	74 1520	9750	-12 8000	13 9520
3000	-79 7000	85 8710	9900	-8 6000	9 5410
4000	-85 2000	91 6300	1 0000	-5 6000	6 2780
5000	-85 4000	91 7540			

RADI OK Cancel HELP

(a)

CROSS SECTION AT RADIUS 300

CHORD FRACTION	BACK OFFSET	FACE OFFSET	CHORD FRACTION	BACK OFFSET	FACE OFFSET
0000	0000	0000	6000	-79 4100	85 1580
0100	-15 8850	17.4060	7000	-69 8420	73 7560
0250	-24 9940	27 1770	8000	-53 8710	56 8030
0500	-35 3260	38 1960	9000	-32 3630	34 4370
1000	-49 8620	53 5820	9500	-19 5970	21 0790
2000	-68 8120	73 5360	9750	-12 7500	13 8650
3000	-79 8940	85 1260	9900	-8 6170	9 4910
4000	-85 4150	90 8190	1 0000	-5 5990	6 2620
5000	-85 6640	90 9130			

RADI OK Cancel HELP

(b)

CROSS SECTION AT RADIUS [illegible]

CHORD FRACTION	BACK OFFSET	FACE OFFSET	CHORD FRACTION	BACK OFFSET	FACE OFFSET
0000	0000	0000	6000	-87 0470	70 6330
0100	-15.6300	15.2510	7000	-72 2950	60 9030
0250	-24 8340	23 5510	8000	-55 8710	46 7870
0500	-35 4140	32 7820	9000	-33 1610	28 8040
1000	-50 4150	45 5410	9500	-19 7750	17 9470
2000	-70 0890	61 9550	9750	-12 6850	12 0010
3000	-81 7110	71 3630	9900	-8 4100	8 3910
4000	-87 6010	75 8680	1 0000	-5 3270	5 6590
5000	-88 1000	75 7010			

RADI | OK | Cancel | HELP

(c)

CROSS SECTION AT RADIUS .400

CHORD FRACTION	BACK OFFSET	FACE OFFSET	CHORD FRACTION	BACK OFFSET	FACE OFFSET
0000	0000	0000	6000	-84 5990	56 9320
0100	-15 1150	13 1660	7000	-73 2330	48 7980
0250	-24 2660	20 0610	8000	-56 6890	37 3580
0500	-34 8800	27 5920	9000	-33 2910	23 4830
1000	-50 0260	37 8880	9500	-19 5950	14 9670
2000	-70 0030	50 9580	9750	-12 4030	10 2210
3000	-81 8980	58 3200	9900	-8 0820	7 3090
4000	-86 0200	61 7310	1 0000	-5 0100	5 0740
5000	-88 7190	61 3280			

RADI | OK | Cancel | HELP

(d)

CROSS SECTION AT RADIUS .500

CHORD FRACTION	BACK OFFSET	FACE OFFSET	CHORD FRACTION	BACK OFFSET	FACE OFFSET
0000	0000	0000	6000	-79 8130	31 5170
0100	-13 0500	9.3350	7000	-70 2100	26 3270
0250	-21 4450	13 6260	8000	-54 5450	19 8630
0500	-31 4100	18 0250	9000	-31 3030	13 5990
1000	-45 8410	23 7120	9500	-17 9050	9 4420
2000	-65 1140	30 5920	.9750	-10 9840	6 9200
3000	-76 7820	34 1600	9900	-6 8530	5 3380
4000	-82 9650	35 5140	1.0000	-3 9830	3 9830
5000	-84 0380	34 6830			

RADI | OK | Cancel | HELP

(e)

[illegible]

CHORD FRACTION	BACK OFFSET	FACE OFFSET	CHORD FRACTION	BACK OFFSET	FACE OFFSET
0000	0000	0000	6000	-74 0670	13 0890
0100	-10 1230	7.6080	7000	-66 0240	12 6860
0250	-16 5700	10 3810	8000	-51 6780	11 0420
0500	-24 7750	12 7020	9000	-30 2230	7 3660
1000	-37.2350	14 3780	9500	-16 6030	4 5300
.2000	-55 1430	14 5070	9750	-9 0110	2 8210
3000	-67 1360	13 7820	9900	-4 1010	1 8860
4000	-74 2130	13 2340	1 0000	- 9030	9030
5000	-76 5660	13 0570			

RADII

(f)

CROSS SECTION AT RADIUS .700

CHORD FRACTION	BACK OFFSET	FACE OFFSET	CHORD FRACTION	BACK OFFSET	FACE OFFSET
0000	0000	0000	6000	-63 3060	- 0180
0100	-7 7440	5 1330	7000	-56 2860	8820
0250	-13 0000	6 5790	8000	-43 8860	1 6580
0500	-19 8790	7 3380	9000	-25 5240	1 7810
1000	-30 6210	6 8790	9500	-13 9350	1 4110
2000	-46 4080	4 1810	9750	-7 5140	1 0760
3000	-57 0970	1 6760	9900	-3 5810	9700
4000	-63 4120	- 0880	1 0000	- 6530	6530
5000	-65 5290	4410			

RADII OK Cancel HELP

(g)

CROSS SECTION AT RADIUS .800

CHORD FRACTION	BACK OFFSET	FACE OFFSET	CHORD FRACTION	BACK OFFSET	FACE OFFSET
0000	0000	0000	6000	-47 9830	-3 6240
0100	-5 5840	3 4510	7000	-42 6070	-2 5490
0250	-9 4860	4 2310	8000	-33 1560	-1.2490
0500	-14 6360	4 4220	9000	-19 2490	- 1040
1000	-22 8030	3 4510	9500	-10 4910	2780
2000	-34 9250	5200	9750	-5 6360	3820
3000	-43 1620	-1 9940	9900	-2 6530	5380
4000	-48 0520	-3 5550	1 0000	- 4510	4510
5000	-49 6820	-4 0750			

RADII

(h)

CROSS-SECTION AT RADIUS [illegible]

CHORD FRACTION	BACK OFFSET	FACE OFFSET	CHORD FRACTION	BACK OFFSET	FACE OFFSET
0000	0000	0000	6000	-27 1880	.5860
0100	-3 3700	2 2850	7000	-24 1750	9000
0250	-5 6410	2 9420	.8000	-18 8630	1 1280
0500	-8 5960	3 3410	9000	-10 9810	1 0000
1000	-13 2090	3 2410	9500	-5 9970	7280
2000	-19 9630	2 2280	9750	-3 2410	5280
3000	-24 5320	1 2570	9900	-1 5420	4570
4000	-27 2310	6430	1 0000	- 2860	2860
5000	-28 1310	4280			

RADII OK Cancel HELP

(i)

CROSS-SECTION AT RADIUS [illegible]

CHORD FRACTION	BACK OFFSET	FACE OFFSET	CHORD FRACTION	BACK OFFSET	FACE OFFSET
0000	0000	0000	6000	-15 6800	4 3330
0100	-2 2650	1 8050	7000	-13 9950	4 0700
0250	-3 6650	2 5170	8000	-10 9750	3 4140
0500	-5 4270	3 1730	9000	-6 4450	2 1880
1000	-8 0530	3 7970	9500	-3 5450	1 3020
2000	-11 7730	4.2130	9750	-1 9370	7770
3000	-14 2460	4 3220	9900	- 9520	4810
4000	-15 7120	4 3660	1 0000	- 2080	2080
5000	-16 1940	4 3770			

RADII OK Cancel HELP

(j)

CROSS-SECTION AT RADIUS .975

CHORD FRACTION	BACK OFFSET	FACE OFFSET	CHORD FRACTION	BACK OFFSET	FACE OFFSET
0000	0000	0000	6000	-9 3590	6 9860
0100	-1 7140	1 6150	7000	-8 4200	6 3440
0250	-2 6450	2 4060	8000	-6 6730	5 0910
0500	-3 7490	3 2790	9000	-3 9710	3 0810
1000	-5 2890	4 3990	9500	-2 2160	1 7470
2000	-7 3240	5 7420	9750	-1 2280	9890
3000	-8 6260	6 5500	9900	- 6340	5360
4000	-9 3840	7 0110	1 0000	- 1650	1650
5000	-9 6390	7 1680			

RADII OK Cancel HELP

(k)

Figure 2.8: Cross-section Data for TRUMP Propeller Blade

Figures 2.9 and 2.10 show plots of the back and face offsets at the first radial position (radial fraction = 0.298).

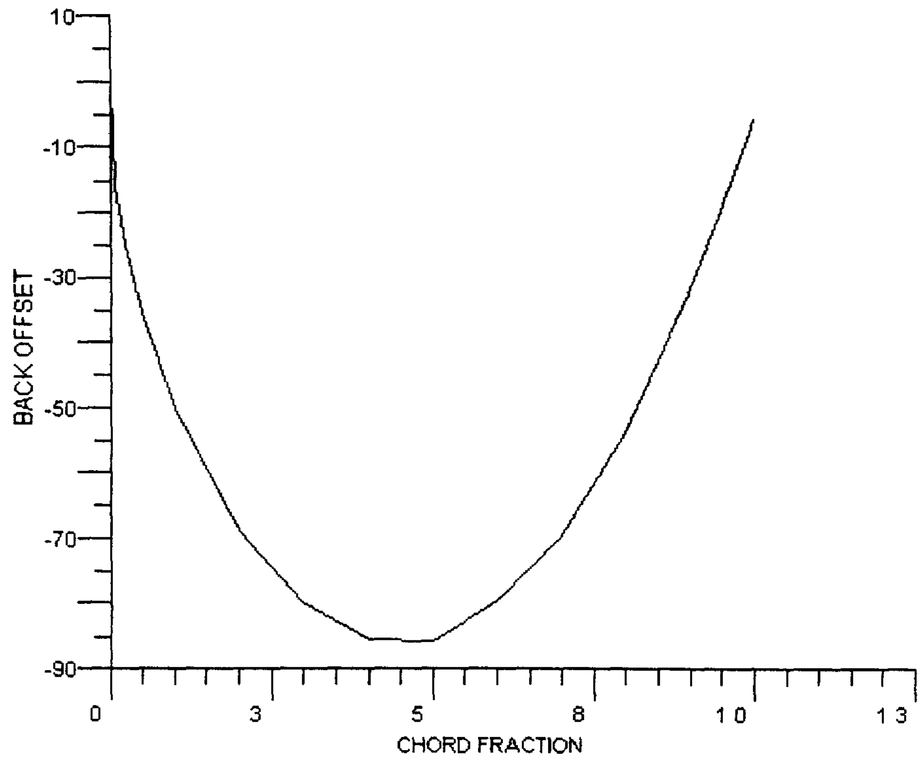


Figure 2.9: Variation of Back Offsets with Chord Fractions for TRUMP Propeller Blade

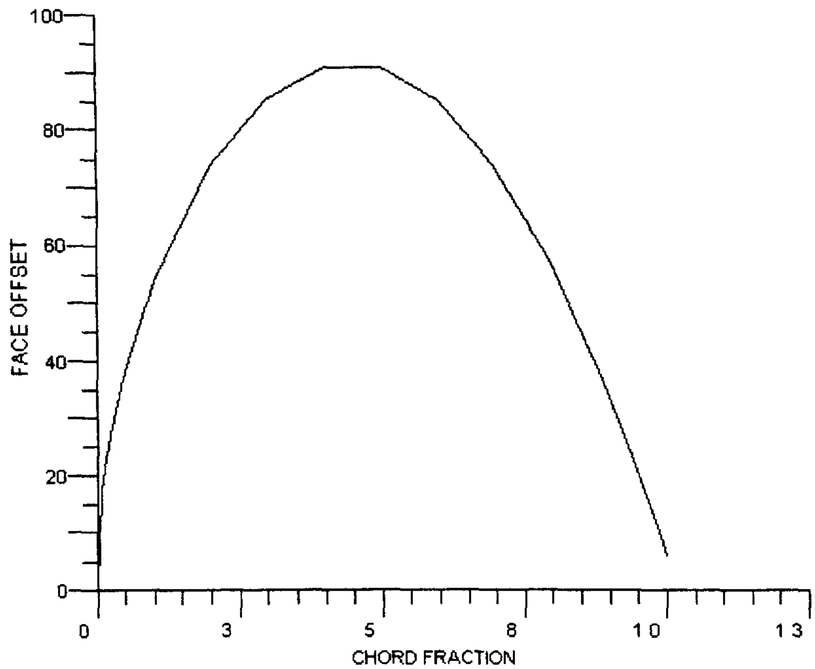


Figure 2.10: Variation of Front Offsets with Chord Fractions for TRUMP Propeller Blade

2.2.4 Edge Data

Figure 2.11 shows the blade edge data entered for the TRUMP blade.

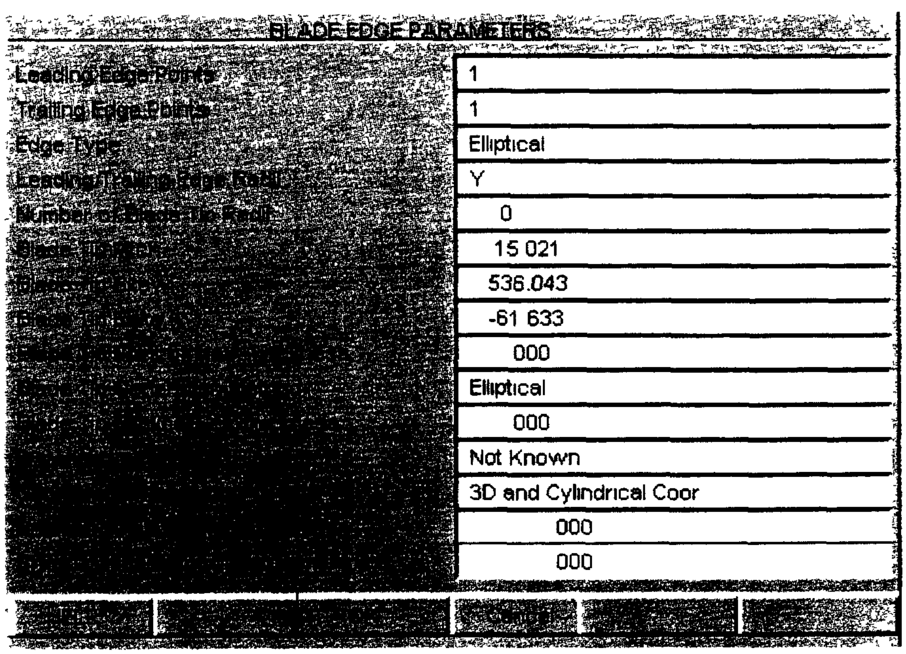
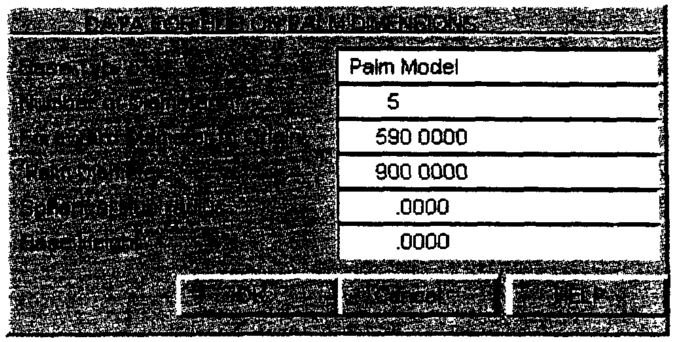


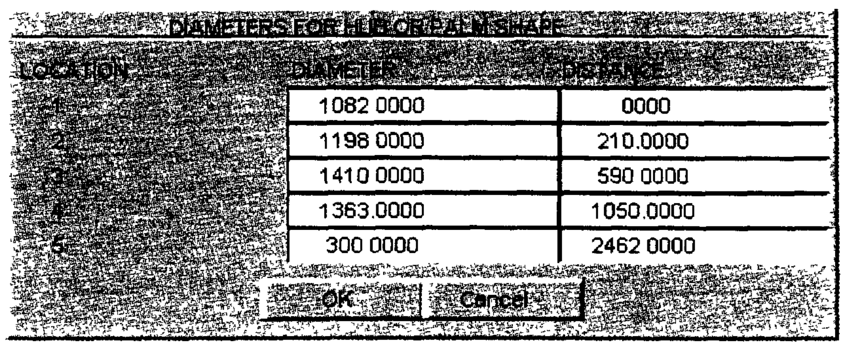
Figure 2.11: TRUMP Blade Edge Parameters

2.2.5 Hub/Palm Data

Figure 2.12 shows the hub/palm data entered for this blade.



(a)



(b)

Figure 2.12: Hub/Palm Data for TRUMP Propeller Blade

This completes the blade geometry data definitions. By clicking **EXIT** on the blade geometry submenu (Figure 2.1), the program generates the complete blade geometry.

2.2.6 Plots of TRUMP Blade Geometry

Several options are provided for plotting the blade geometry including the blade data. These options are available under the **Special Plots (Model)** option and are accessed by clicking **Display** on the PVAST main menu, **Special Plots (Model)** and then **Blade Geometry**. The available options are shown in Figure 2.13.

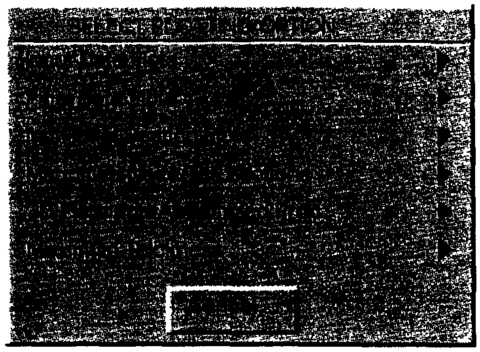
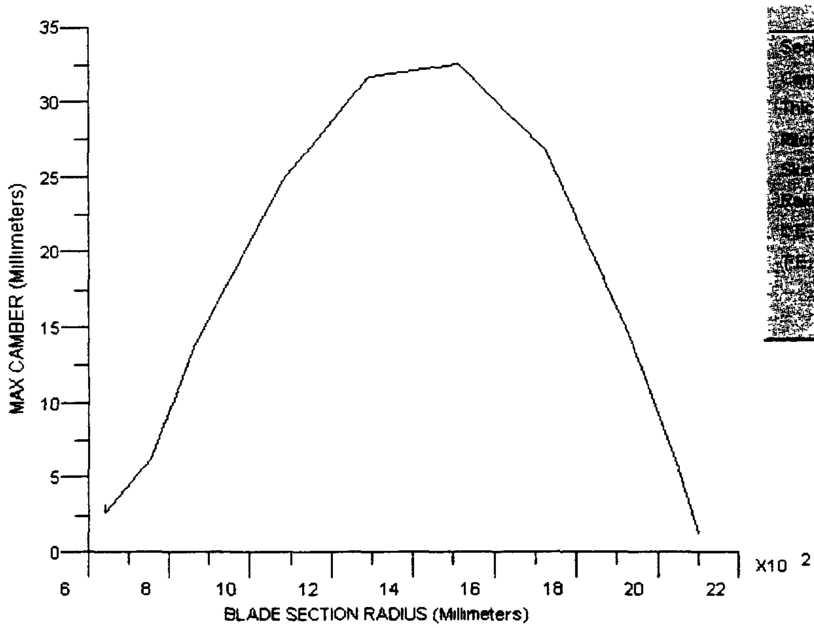


Figure 2.13: Blade Geometry Plotting Options

Final Data Plots

Figures 2.14 and 2.15 show plots of the maximum camber and maximum thickness versus the section radii plotted through the **Final Data Plots** option (Figure 2.13).



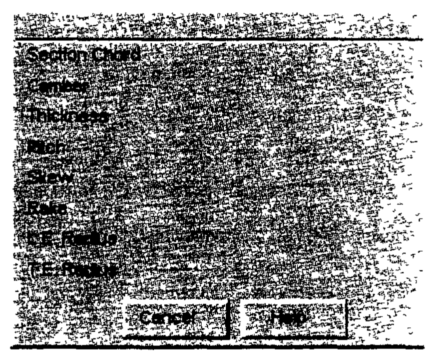
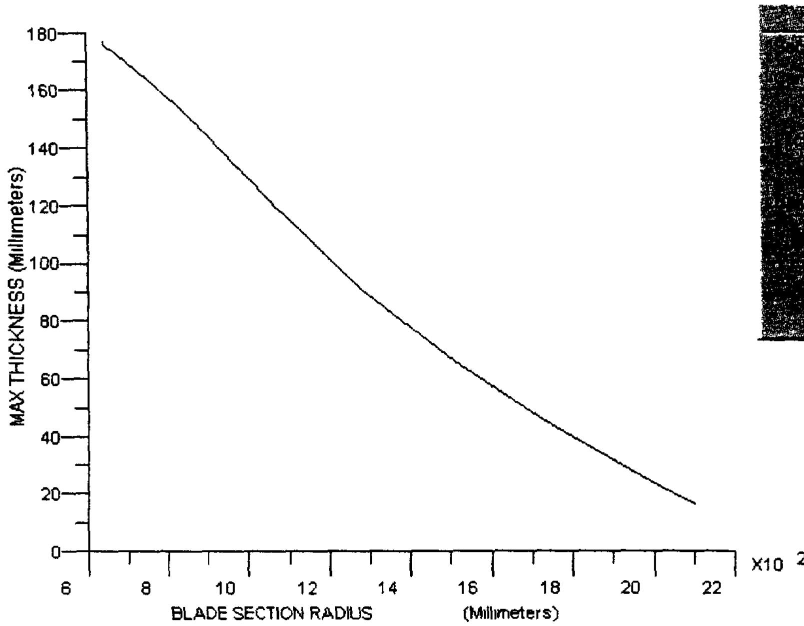


Figure 2.14: Maximum Camber vs Section Radius of TRUMP Blade



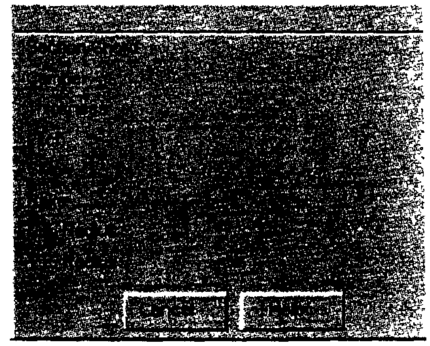


Figure 2.15: Maximum Thickness vs Section Radius of TRUMP Blade

Individual Section Plots

Figure 2.16 shows a typical section plot (at radial fraction 0.4) plotted through the **Individual Section Plots** option (Figure 2.13).

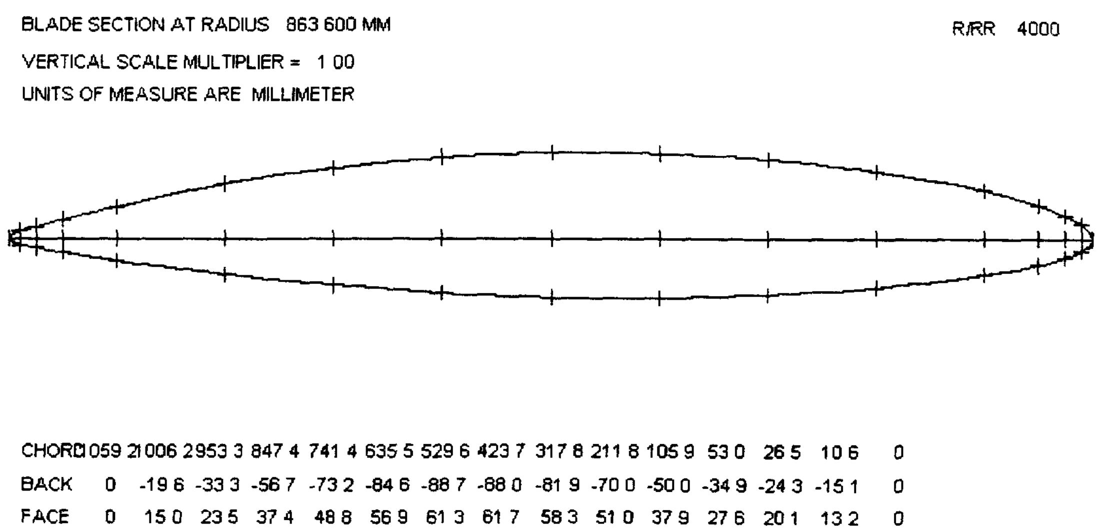


Figure 2.16: Individual Section Plot for TRUMP Propeller Blade

Magnified Nose/Tail Sections

Figure 2.17 shows the magnified leading edge section (at 0.4 radial fraction) plotted through the **Magnified Nose/Tail Sections** plotting option (Figure 2.13).

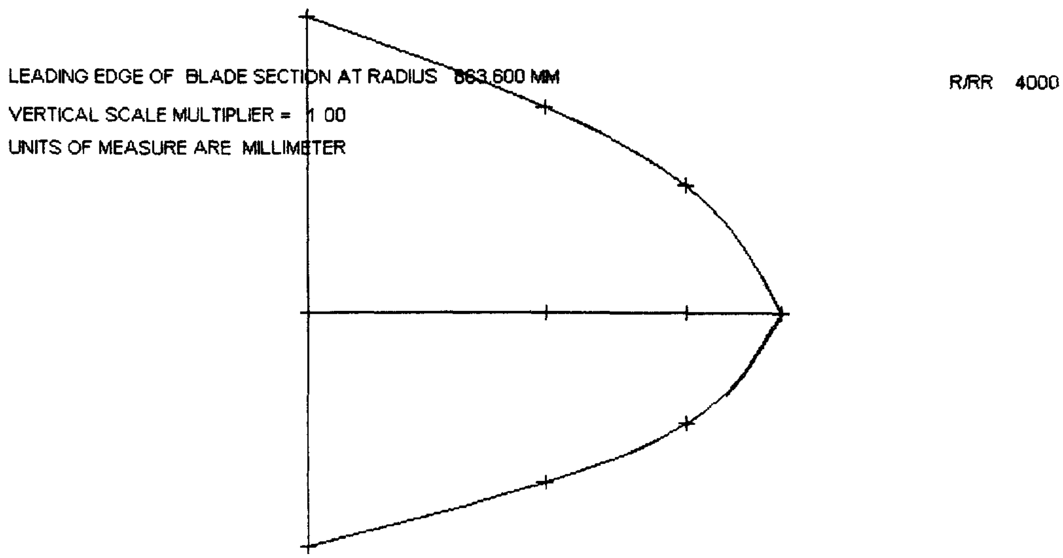


Figure 2.17: Magnified Leading Edge of TRUMP Propeller Blade

3-D Plots and Assembled Prop

Figures 2.18 to 2.28 show views of a single blade of the TRUMP propeller.

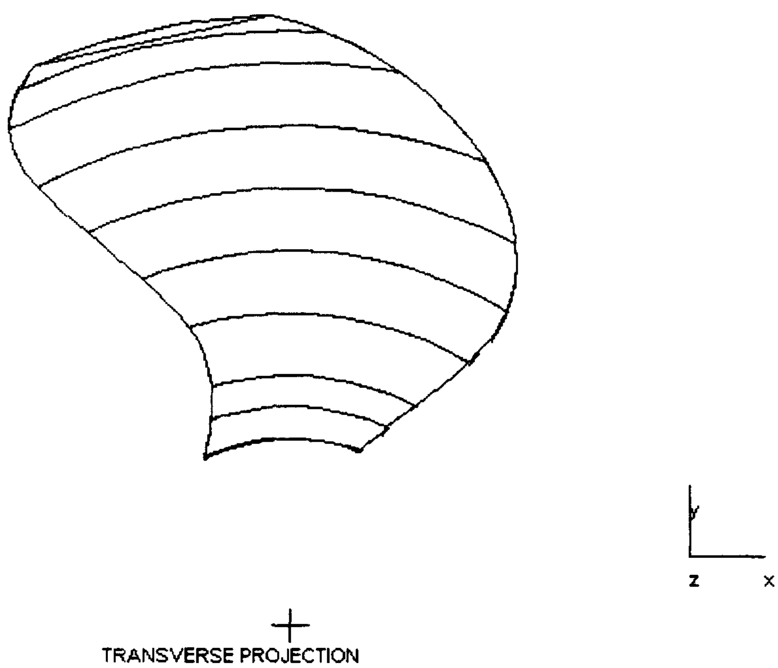


Figure 2.18: Transverse Projection of TRUMP Propeller Blade

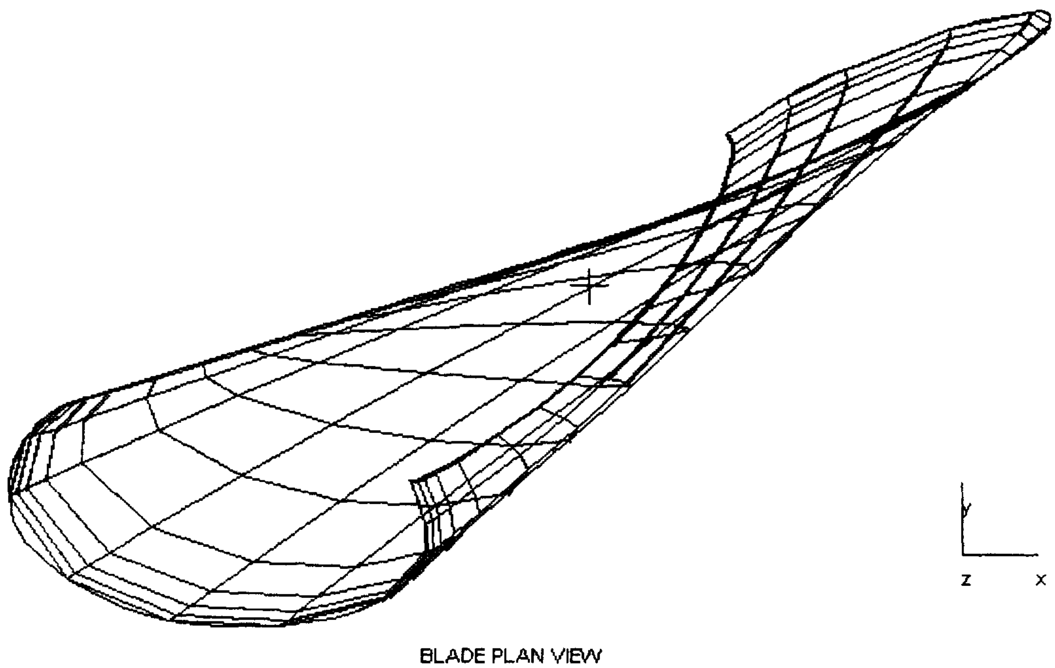


Figure 2.19: Plan View of TRUMP Propeller Blade

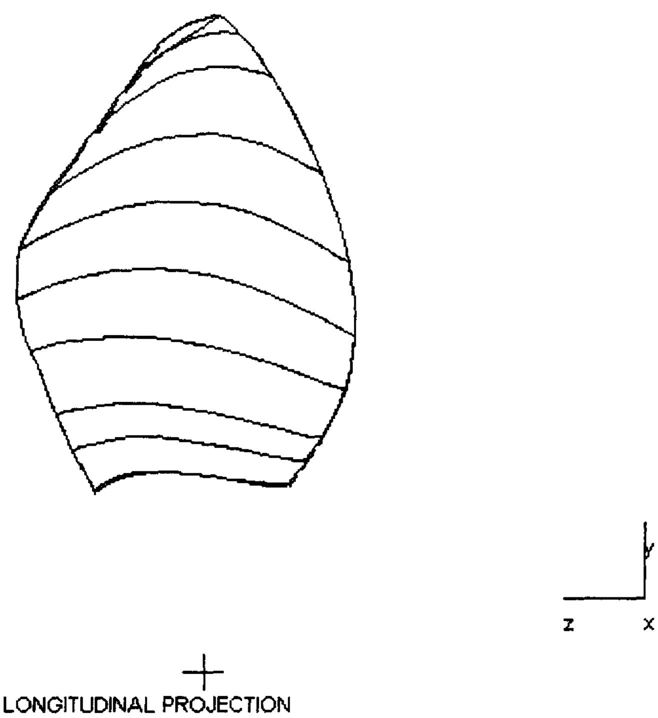


Figure 2.20: Longitudinal Projection of TRUMP Propeller Blade

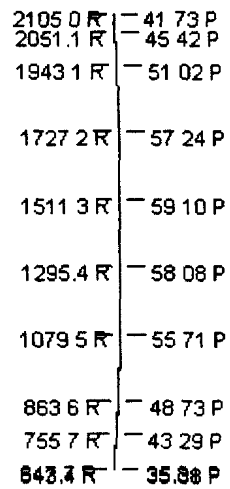


Figure 2.21: Pitch Diagram of TRUMP Propeller Blade

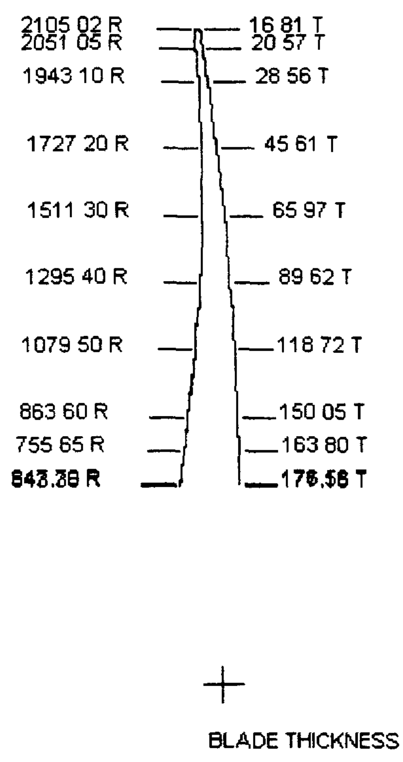


Figure 2.22: Blade Thickness at 50% Chord for TRUMP Propeller Blade

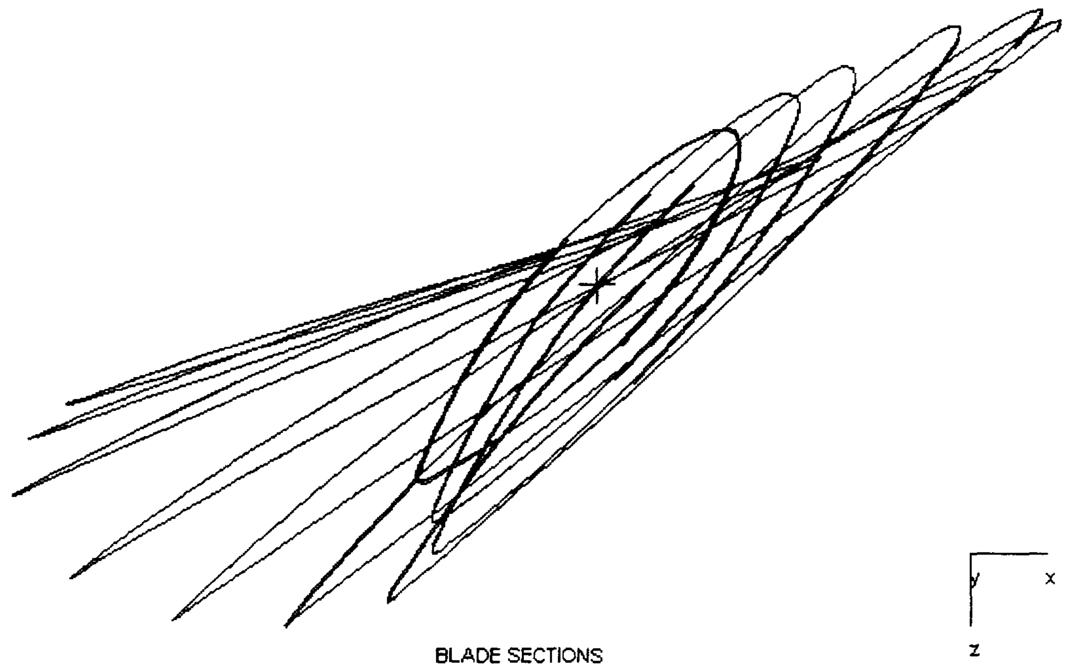


Figure 2.23: Plan View of Blade Sections for TRUMP Propeller Blade

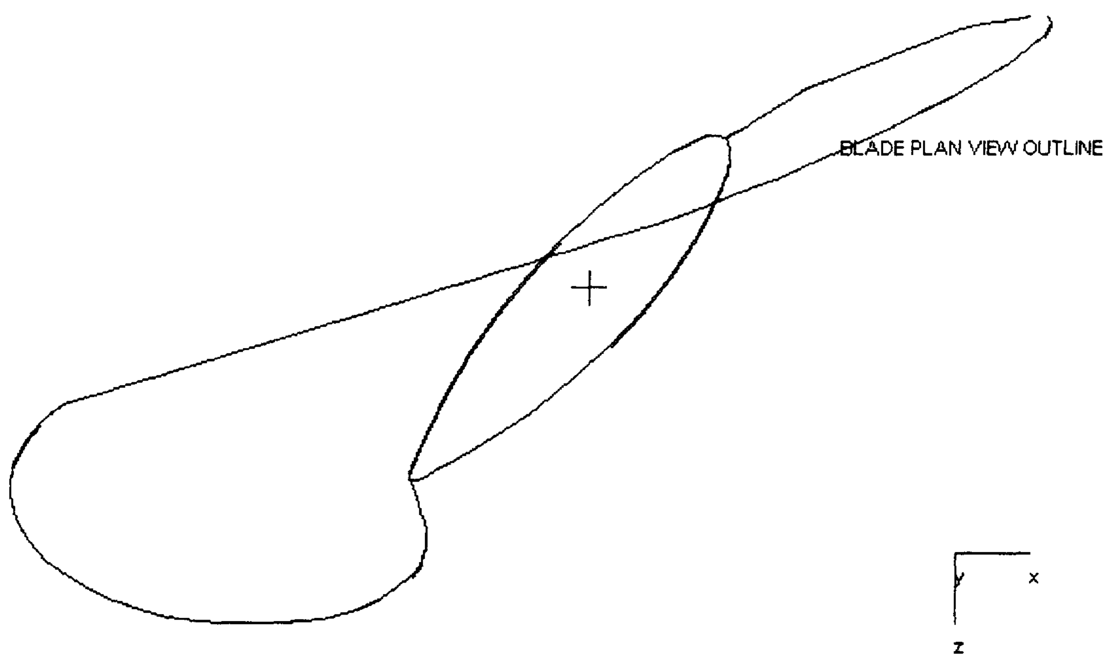
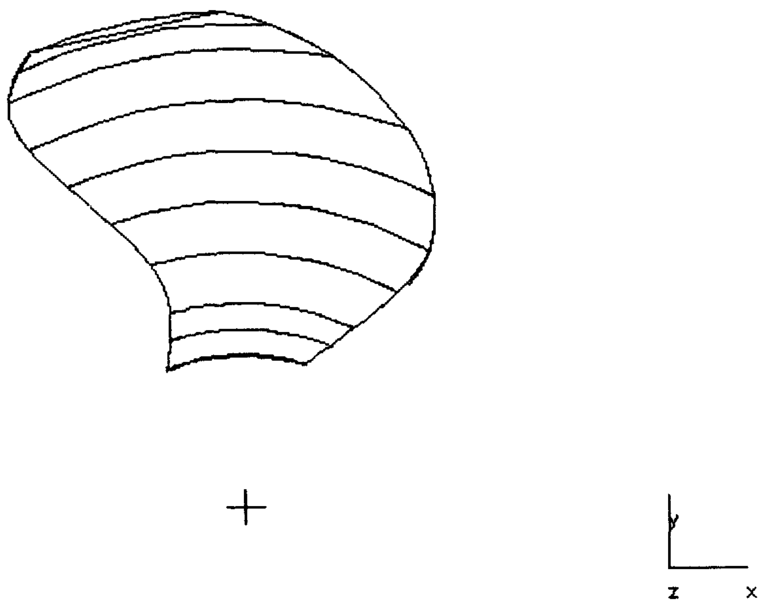
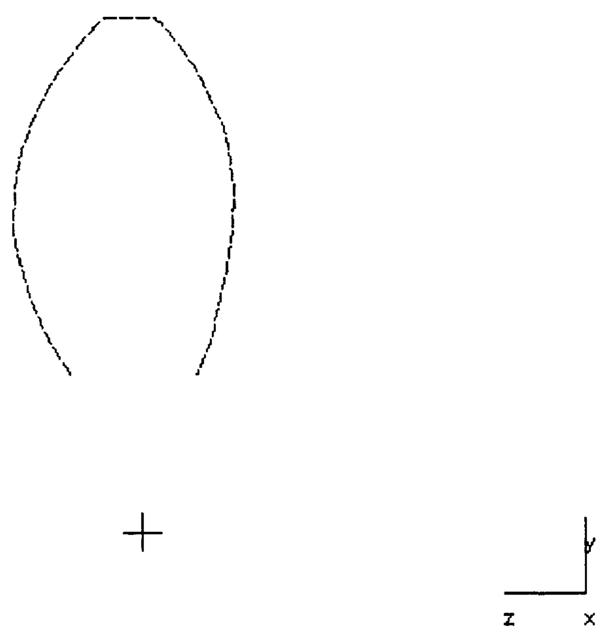


Figure 2.24: Plan View Outline of TRUMP Propeller Blade



DEVELOPED OUTLINE

Figure 2.25: Developed Outline of TRUMP Propeller Blade



SWEPT AREA

Figure 2.26: Swept Area of TRUMP Propeller Blade

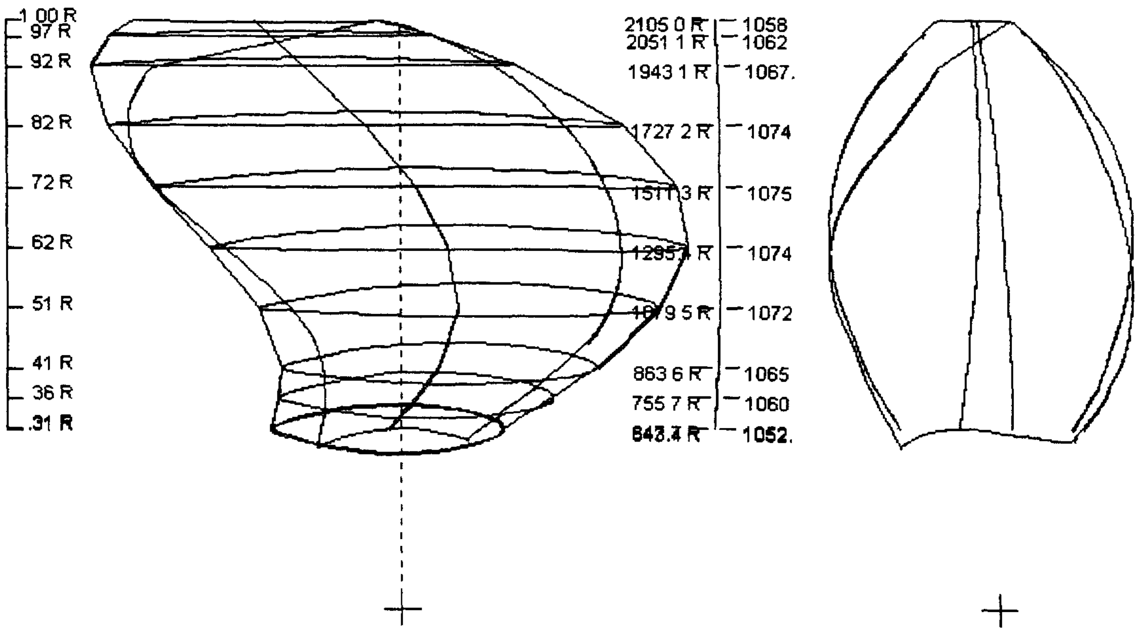


Figure 2.27: Combined Drawing of TRUMP Propeller Blade (3rd Angle Projection)

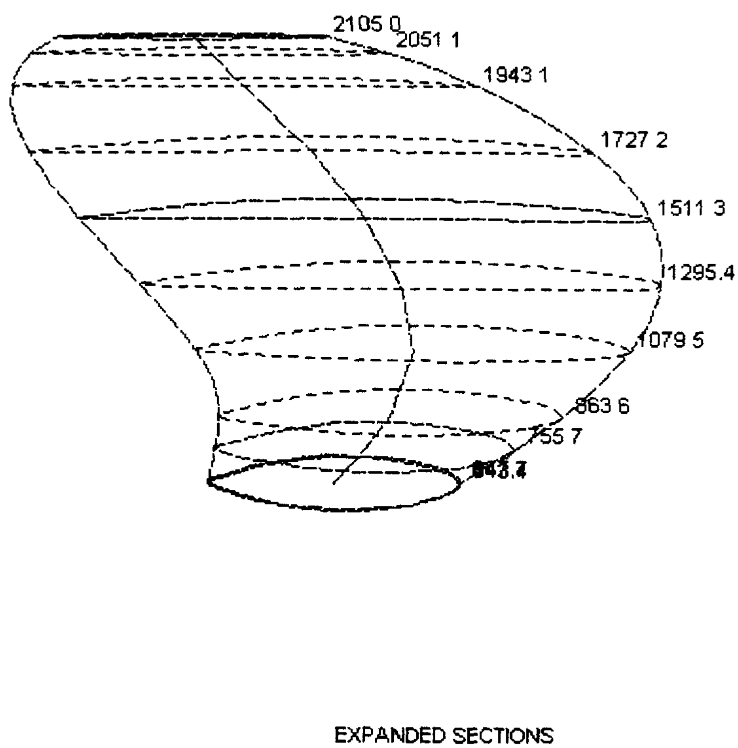


Figure 2.28: Expanded View of TRUMP Propeller Blade

Figure 2.29 shows a view of an assembled propeller with 3 blades.

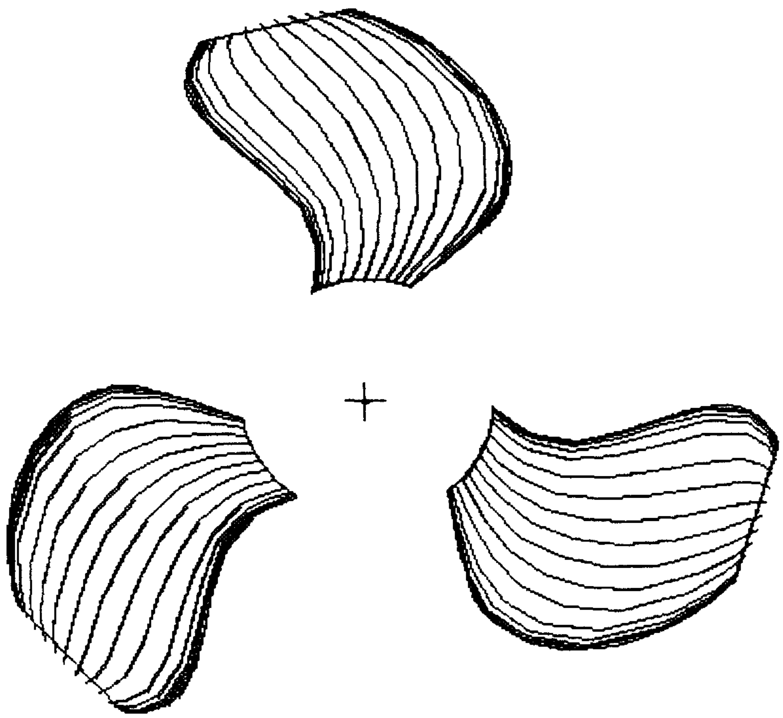
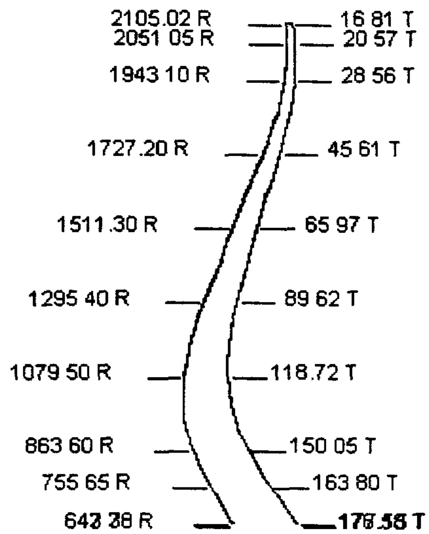


Figure 2.29: View of Assembled TRUMP Propeller Blade

Maximum Thickness

Figure 2.30 shows plots of the maximum thickness with and without rake, plotted through the **Maximum Thickness** plotting option.



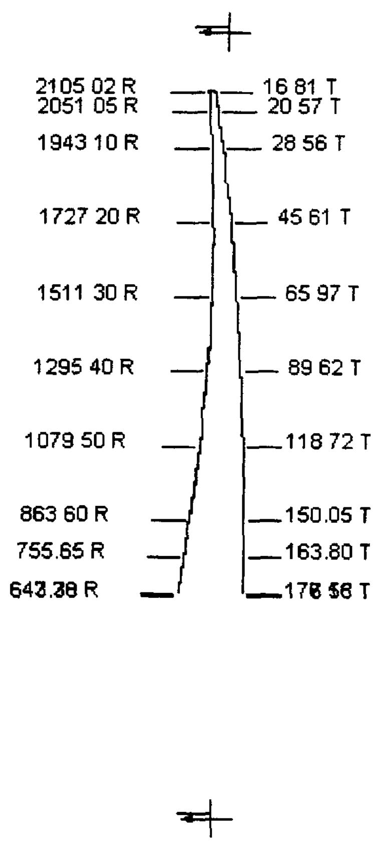


Figure 2.30: Maximum Thickness of TRUMP Propeller Blade With and Without Rake

Following the above proceedings , the geometries of other blades can be defined and plotted.

2.3 Geometry of NRC45 Propeller Blade

The blade geometry data for this propeller blade is shown in the NRC45.DAT file presented in Appendix A.

2.4 Geometry of P5363 Propeller Blade

The blade geometry data for this propeller blade is shown in the P5363.DAT file presented in Appendix A.

2.5 Geometry of P4388 Propeller Blade

The blade geometry data for the P4388 propeller blade is shown in the P4388.DAT file presented in Appendix A.

The blade geometry definition step generates PREFIX.T16 files which are used for developing models in Chapters 3 to 9.

3. BLADE ONLY MODEL FOR VIBRATION ANALYSIS IN AIR

3.1 Introduction

In this chapter, we shall demonstrate how to generate a propeller blade finite element model, perform a natural frequency analysis of the model; and to view the results. It is assumed that the blade geometry data has been entered and geometry defined, as described in Chapter 2. Thus, a PREFIX.T16 file is now available in the working directory, where, "PREFIX" is the alphanumeric prefix of the file name. The narrow blade (NRC45) propeller blade shall be used for this demonstration. With the blade geometry already defined, the remaining steps are:

(i) To generate the finite element model
(ii) To perform the finite element analysis using the VAST program; and
(iii) To post process and view the results

These steps are described in Sections 3.2 to 3.4.

In Sections 3.5 and 3.6 the generation of *Blade Only* finite element models for two other blades, namely the P5363 and P4388 propeller blades, respectively, are presented.

3.2 Finite Element Model Generation

This step involves the development of the finite element model of the propeller blade. This step involves the following sub-tasks:

- Selection of model type;
- Generation of FE grid proportions;
- Generation of the blade FE mesh; and
- Generation of boundary conditions.

These subtasks are described in the following subsections.

3.2.1 Select Model Type

Click **Model** on the main menu and ensure that the **Blade Only** model is highlighted, as shown in Figure 3.1.

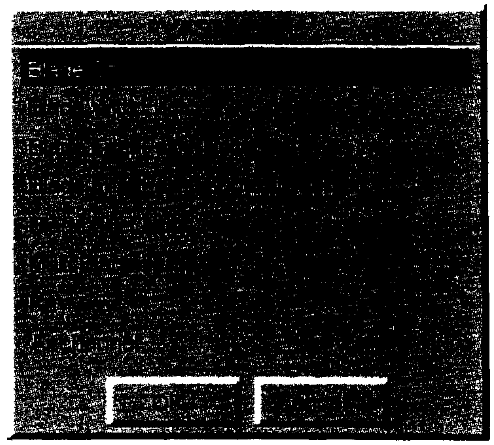


Figure 3.1: Model Submenu

3.2.2 Generate Finite Element Grid Proportions

Click the **Create/Modify** on the main menu and select the **FE Grid Proportions** option on the **CREATE/MODIFY** submenu (Figure 3.2).

The **FE Grid Proportions** submenu appears as shown in Figure 3.3. This submenu also provides capability for editing the blade geometry data available on the PREFIX.T16.

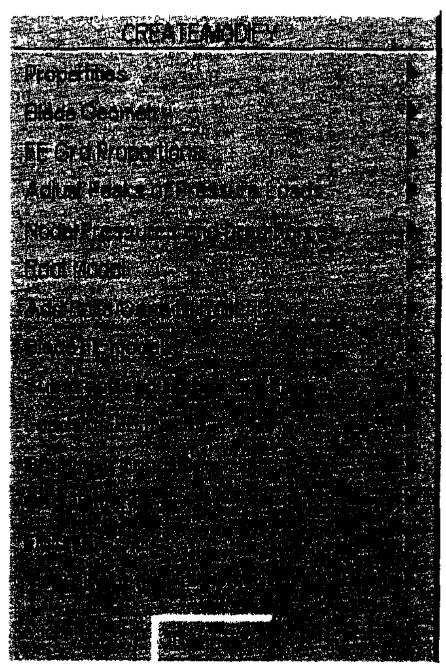


Figure 3.2: Create/Modify Submenu

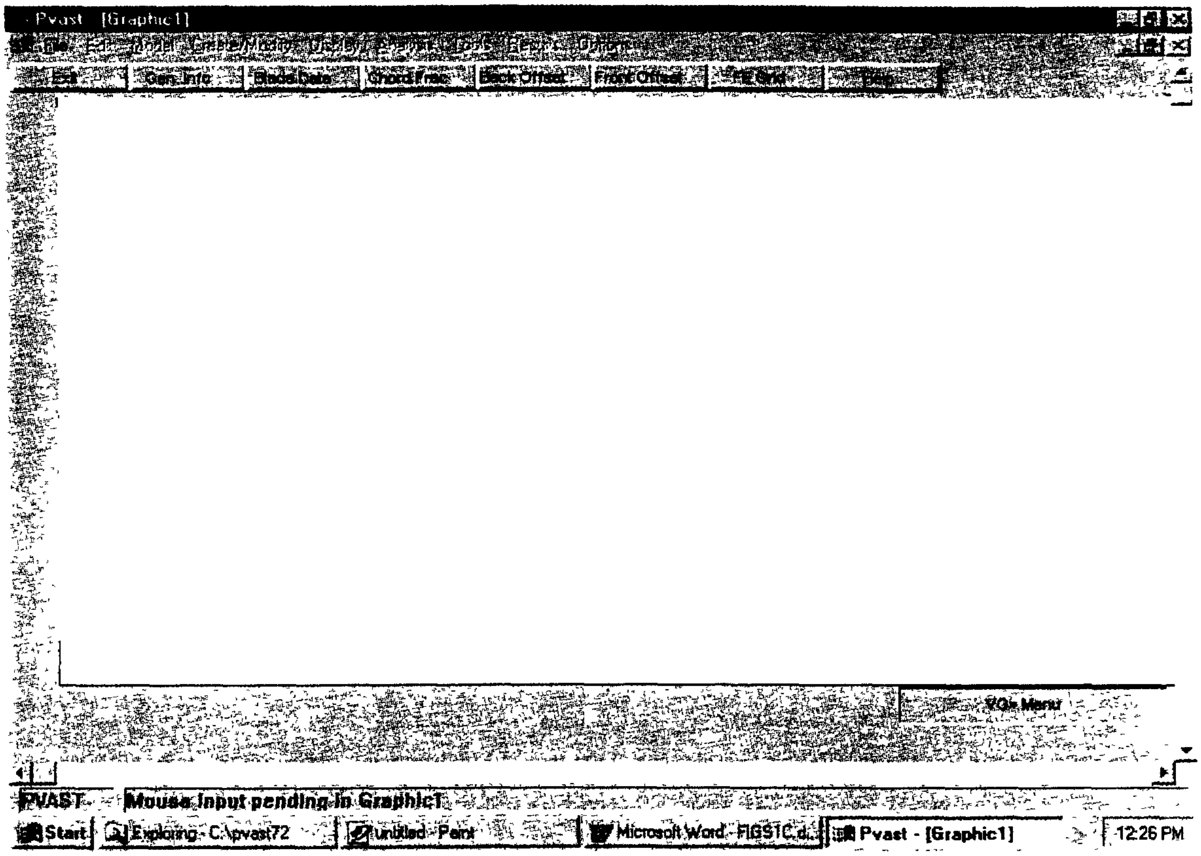
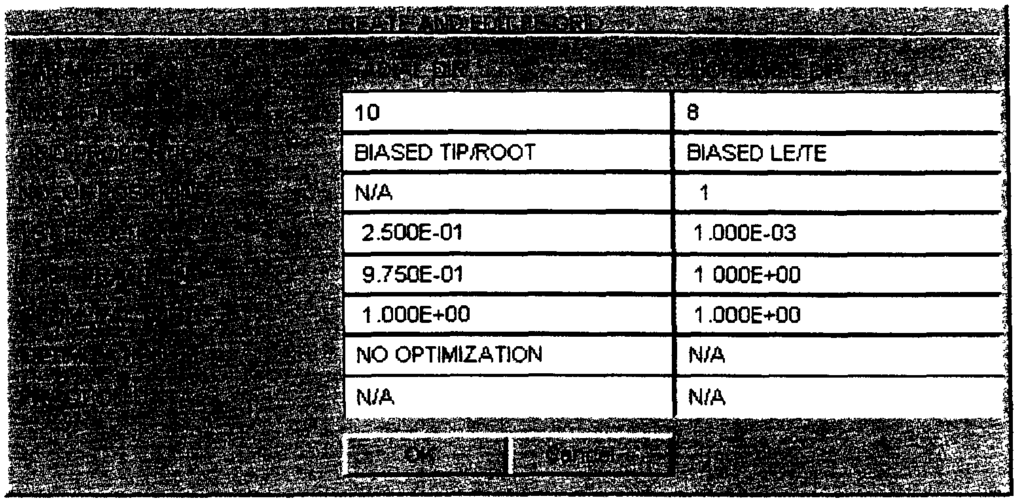


Figure 3.3: FE Grid Proportions Submenu

If it is of interest to view the data, as in the case when the PREFIX.T16 file is the starting point of the modeling, click on

Gen. Info: To view general information about the blade data
Blade Data To view the radii, chord, pitch, skew and rake data
Chord Frac: To view chord fraction data
Back Offset: To view back offset data
Front Offset: To view front offset data

If the PREFIX.T16 file is not the starting point the above reviews may not be necessary and you should go directly and click **FE Grid**. Figure 3.4 shows the grid proportion data entered for this analysis.



10	8
BIASED TIP/ROOT	BIASED LE/TE
N/A	1
2.500E-01	1.000E-03
9.750E-01	1 000E+00
1.000E+00	1.000E+00
NO OPTIMIZATION	N/A
N/A	N/A

Figure 3.4: Edit Table for FE Grid Proportion

Click **OK** (Figure 3.4), when finished
Click **Exit** on the **FE Grid Proportion** submenu (Figure 3.3).
The FE grid proportions are now generated and a message is displayed at the bottom left corner of the PVAST window.

To display the FE grid proportions
Click **Display** on the main menu
Click **Special Plots** on the **MODEL DISPLAY** submenu
Click Blade Grid Proportions on the Special Plots (Model) submenu
Then select **Grid Proportions**, as shown in Figure 3.5

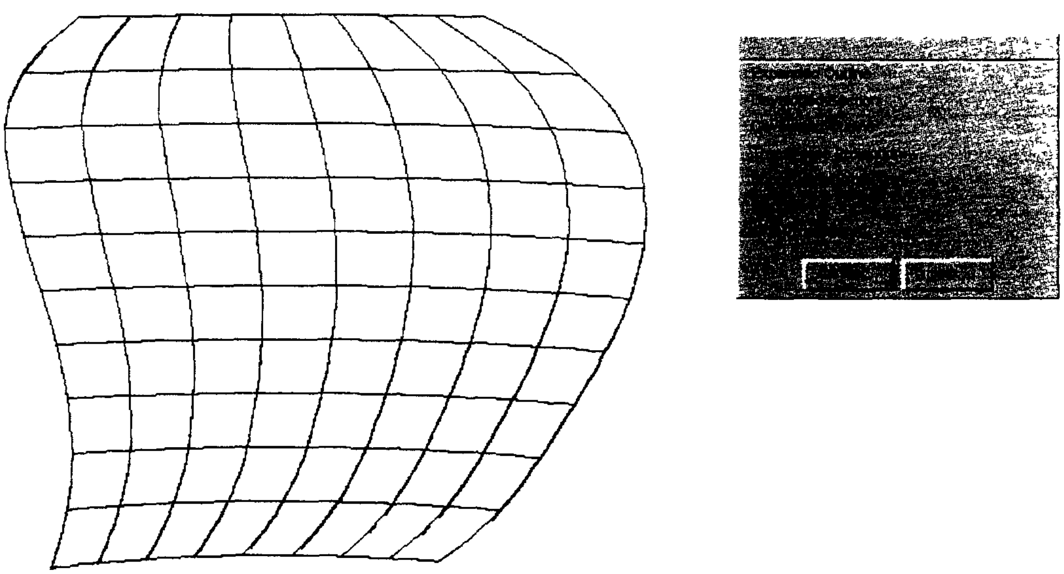


Figure 3.5: FE Grid Proportion for NRC45 Propeller Blade

3.2.3 Generate Finite Element Model

Click the **Create/Modify** on the main menu and select the **FE Model** option on the **CREATE/MODIFY** submenu (Figure 3.2).

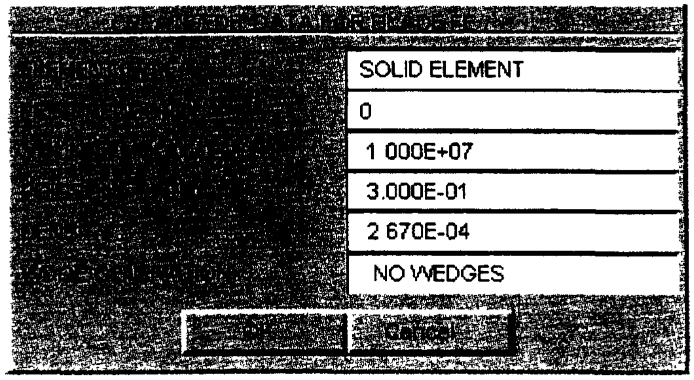


Figure 3.6: Edit Table for Finite Element Model Data

Figure 3.6 shows the edit table for entering the finite element model data. In this example, solid elements have been selected to model the blade, and no wedges are formed at the leading and trailing edges of the blade.

Click **OK** (Figure 3.6), when finished
The FE model is now generated and a message is displayed at the bottom left corner of the PVAST window.

To display the FE model:
Click **Display** on the main menu
Click **Blade FE Model** on the **MODEL DISPLAY** submenu

Then the blade finite element model is displayed as shown in Figure 3.7, using the general purpose plotting capability.

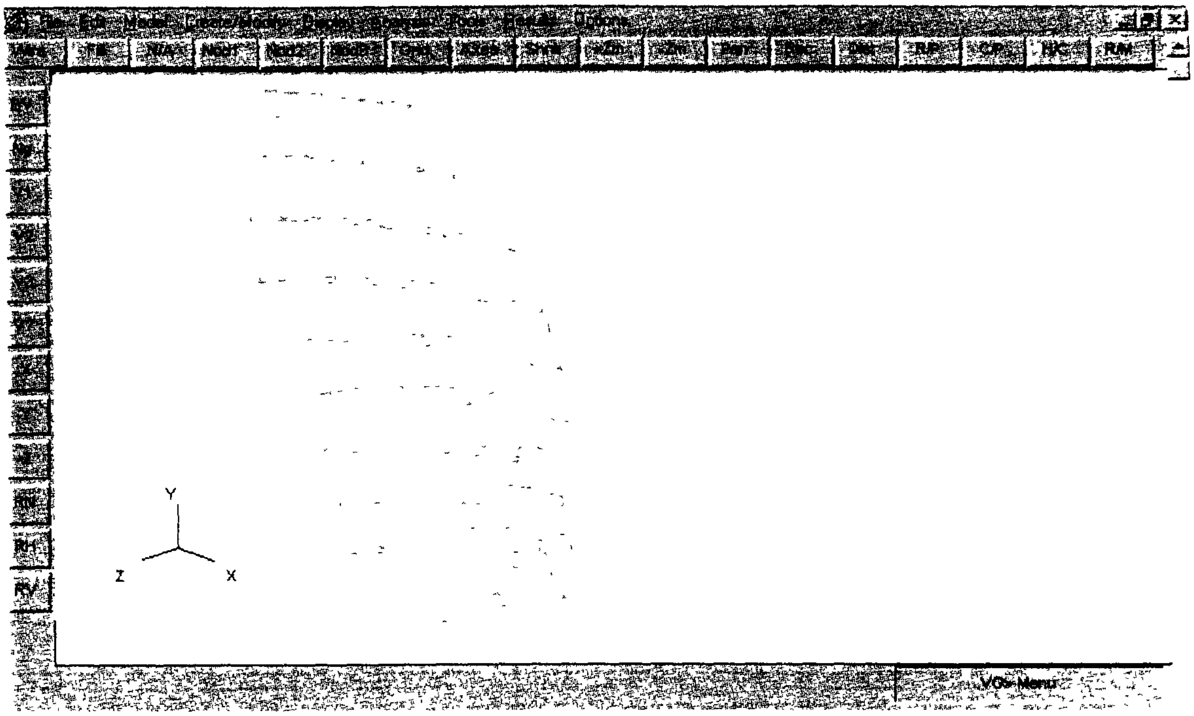


Figure 3.7: Finite Element Model of NRC45 Propeller Blade Using General Plotting Capability

In Figure 3.7, the finite element model has been plotted using the wire mesh plotting option. A fill plot can be made by clicking the **Fill** button near the top right corner. Similarly, other views can be plotted by clicking on **V1**, **V2**, **V3**, **X**, **Y**, **Z**, etc. For a complete description of the general display capabilities, see the MGDSA manual [2].

3.2.4 Generate Boundary Conditions

With the blade finite element model displayed as shown in Figure 3.7,

Click the **Create/Modify** on the main menu and select the **Boundary Conditions** option on the **CREATE/MODIFY** submenu (Figure 3.2).

At this point the user is required to respond to question: ***Should default boundary conditions serve as starting point?*** The default boundary conditions are automatically generated during the finite element model generation. These are usually reasonable boundary conditions, and in most cases they are all the boundary conditions needed for the model. For instance, for the **Blade Only** model, all the nodes at the base of the blade are fixed automatically.

Click **Yes** to accept the default boundary conditions as starting point for the boundary condition definition. Then the default boundary conditions will be displayed as shown in Figure 3.8.

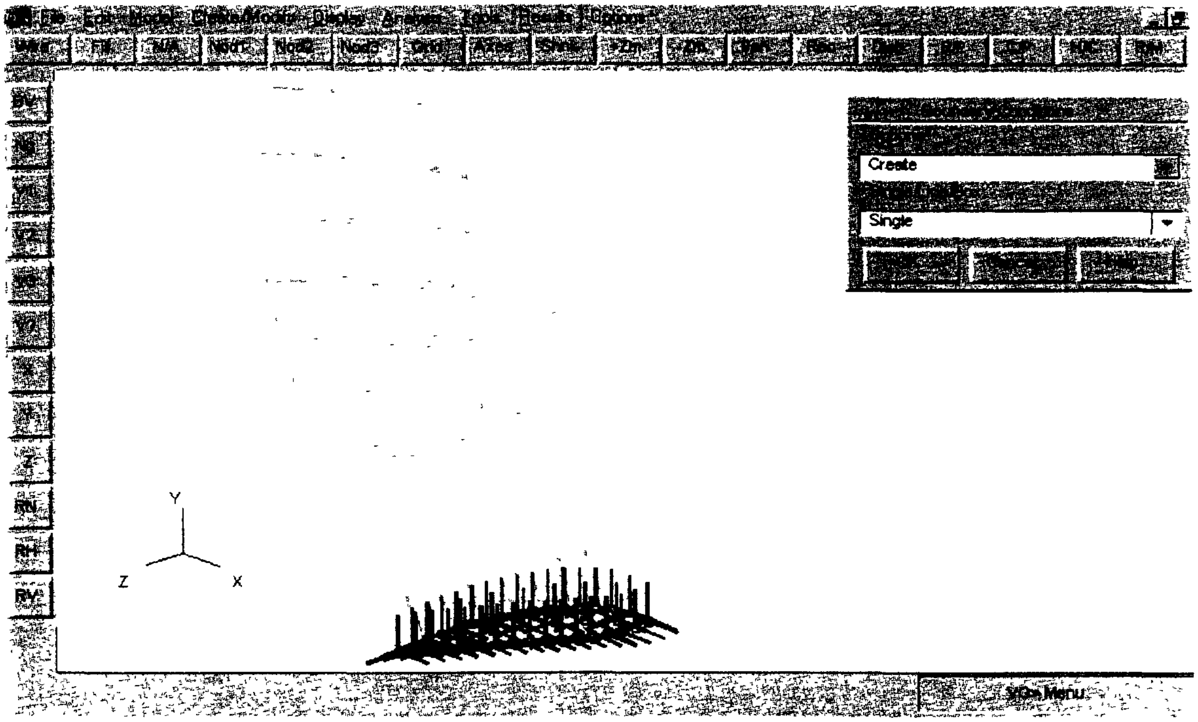


Figure 3.8: Default Boundary Conditions for Blade Only Model

If it is desired to add or edit any of the existing boundary conditions, click on the appropriate boxes on the **Boundary Conditions** submenu, using single, line or box selection of nodes, shown in Figure 3.8.

For the present model, the default boundary conditions shown in Figure 3.8 are the required boundary conditions. So click **Cancel** to end the boundary conditions generation option. The boundary conditions are now defined.

3.3 VAST Finite Element Analysis

This step involves the performance of the finite element analysis using the VAST finite element code. This is accomplished by the following processes:

3.3.1 Generate VAST Input File

This step involves generating the input file for a VAST finite element analysis.

Click the **Create/Modify** on the main menu and select the **VAST Input File** option on the **CREATE/MODIFY** submenu (Figure 3.2).

Select **Natural Frequency** analysis as shown in Figure 3.9. Click **OK**.

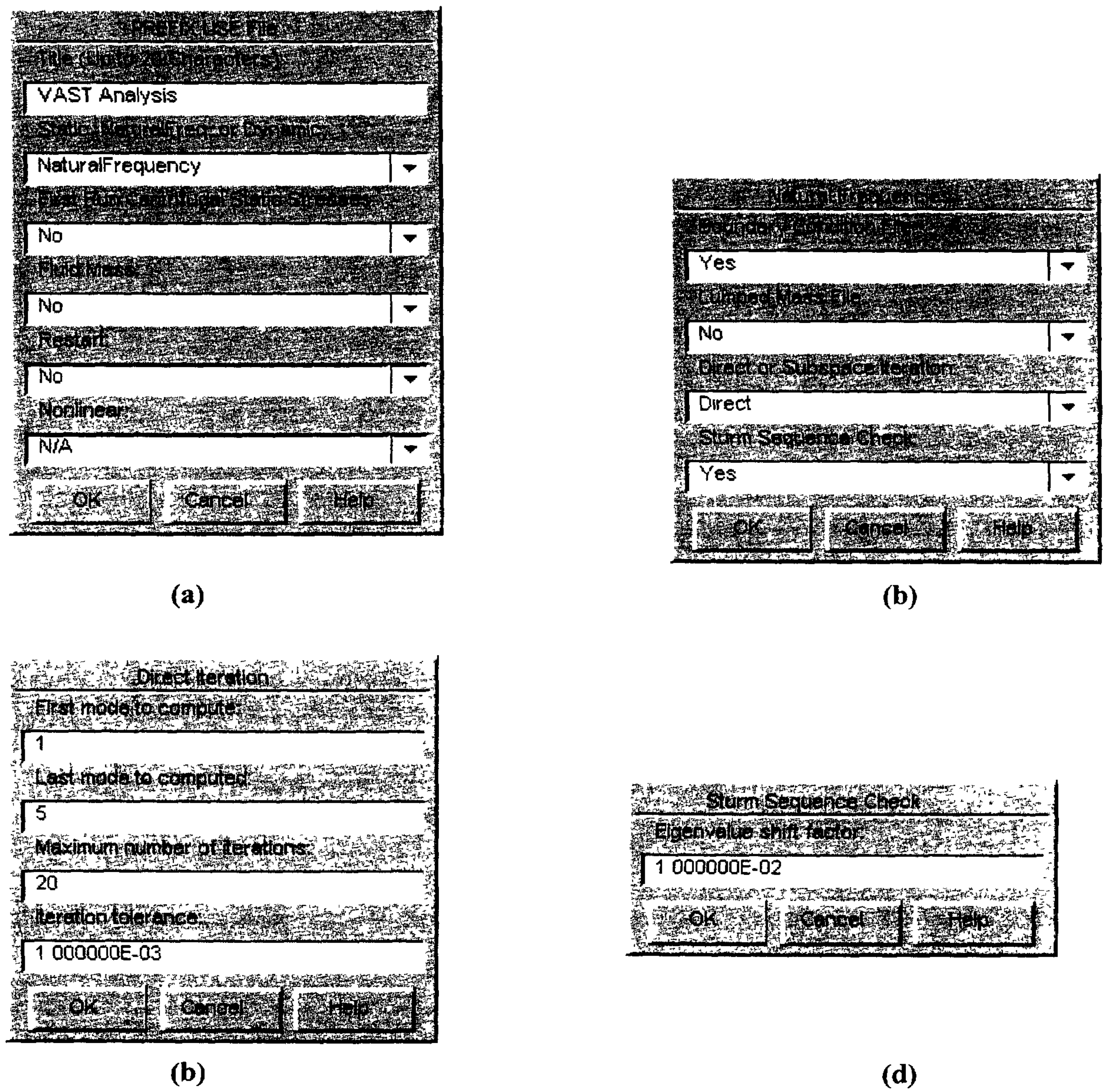


Figure 3.9: Generating VAST Input File for Natural Frequency Analysis

3.3.2 Natural Frequency Analysis

Click **Analysis** on the main menu. Then Click **Start VAST Analysis** on the **Analysis** submenu as shown in Figure 3.10. When the analysis has completed, enter a carriage return to return to the PVAST main menu.

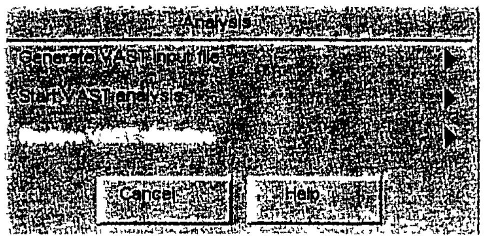


Figure 3.10: Start VAST Analysis

3.3.3 Post Processing of Results

To plot the dry mode results using the general plotting capability, the following steps should be followed:
Click **Results** on the main menu.
Click **Deformation** on the **Results** submenu.
Click **Vibration** mode.

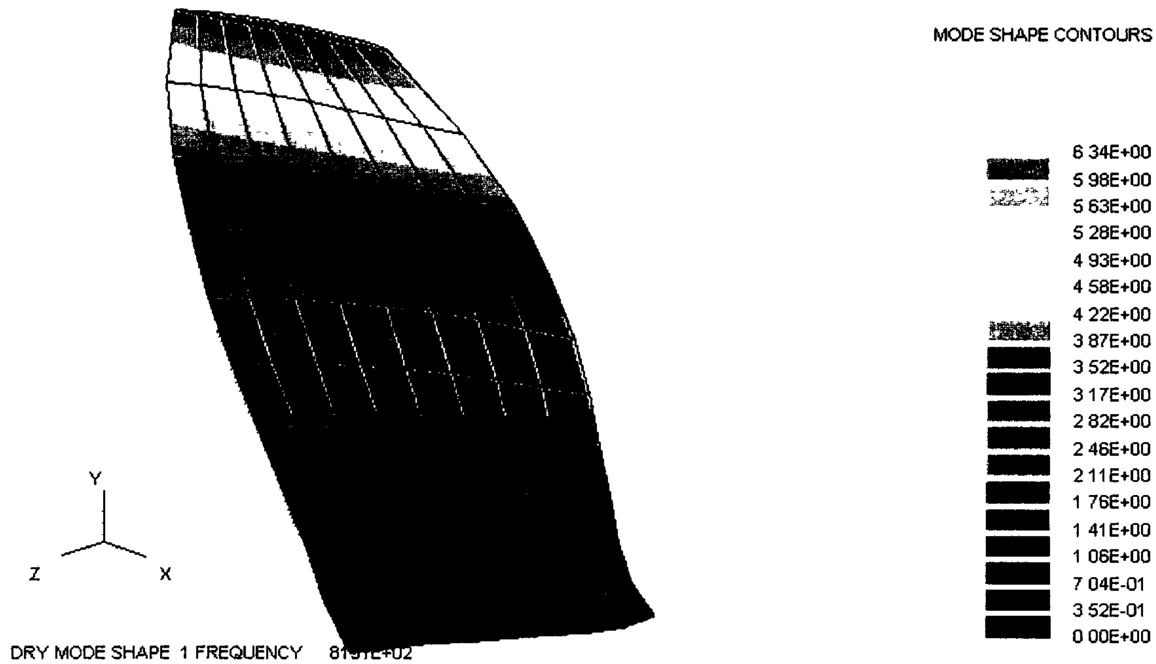
Enter the desired mode number. Now...

Click **Options** on the main menu.
Click **Plotting** on the **Options** submenu.
Click **Contour** on the **Plotting** submenu.
Click **Deformation** on the **Contour** submenu
Select **Fringe** Plotting.

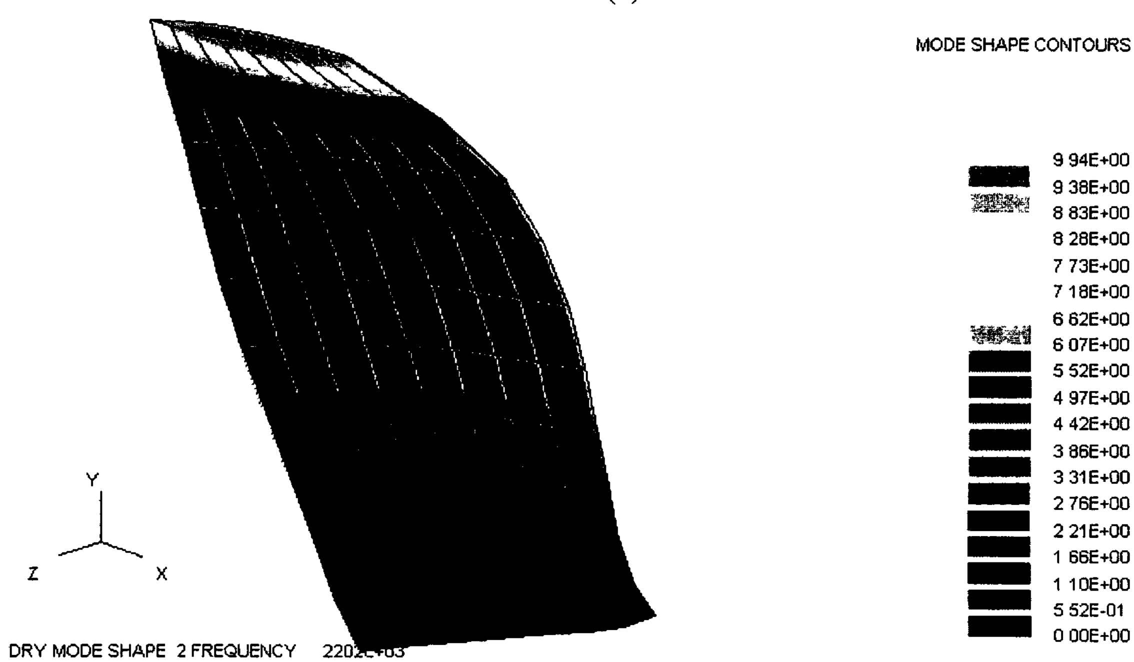
Figures 3.11(a) – (c) show the contours of mode shapes 1-3, respectively.

The capability of plot using the special plotting option is available for dry mode results.

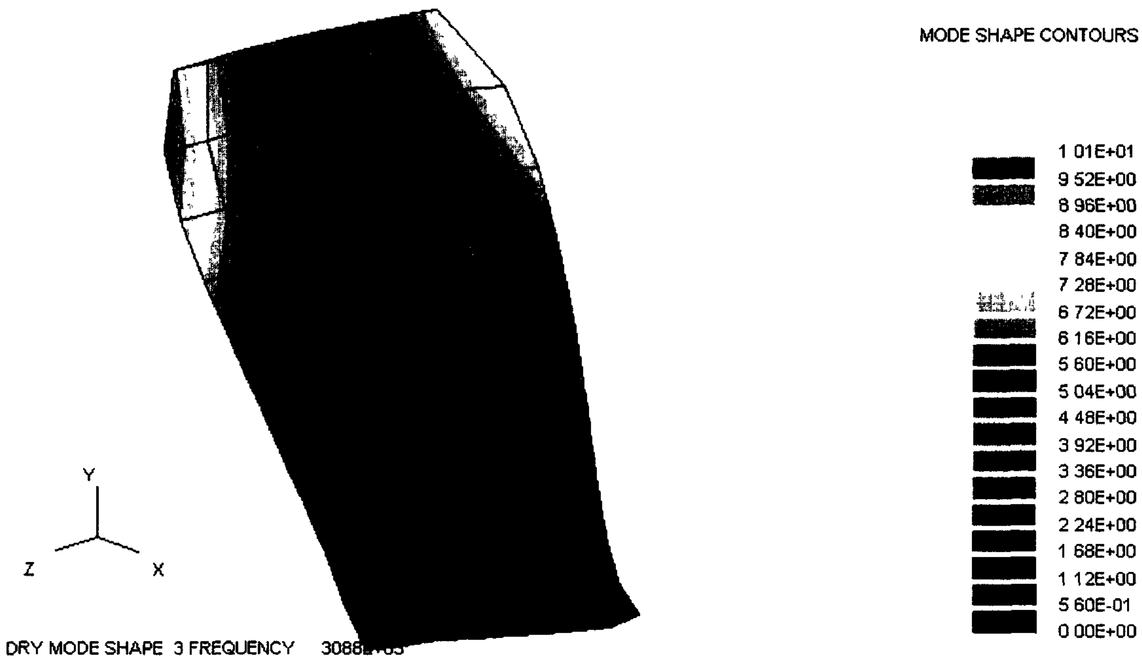
Click **Results** on the main menu.
Click **Special Plots**.
Click **Blade Section Mode Shape.**
Select a radial position.


MODE SHAPE CONTOURS
6 34E+00
5 98E+00
5 63E+00
5 28E+00
4 93E+00
4 58E+00
4 22E+00
3 87E+00
3 52E+00
3 17E+00
2 82E+00
2 46E+00
2 11E+00
1 76E+00
1 41E+00
1 06E+00
7 04E-01
3 52E-01
0 00E+00
Y
Z
X
DRY MODE SHAPE 1 FREQUENCY

(a)


MODE SHAPE CONTOURS
9 94E+00
9 38E+00
8 83E+00
8 28E+00
7 73E+00
7 18E+00
6 62E+00
6 07E+00
5 52E+00
4 97E+00
4 42E+00
3 86E+00
3 31E+00
2 76E+00
2 21E+00
1 66E+00
1 10E+00
5 52E-01
0 00E+00
Y
Z
X
DRY MODE SHAPE 2 FREQUENCY

(b)



(c)

Figure 3.11: Contours of Mode Shapes of NRC45 Propeller in Air: (a) Mode 1; (b) Mode 2; and (c) Mode 3

The resulting plot is shown in Figure 3.12.

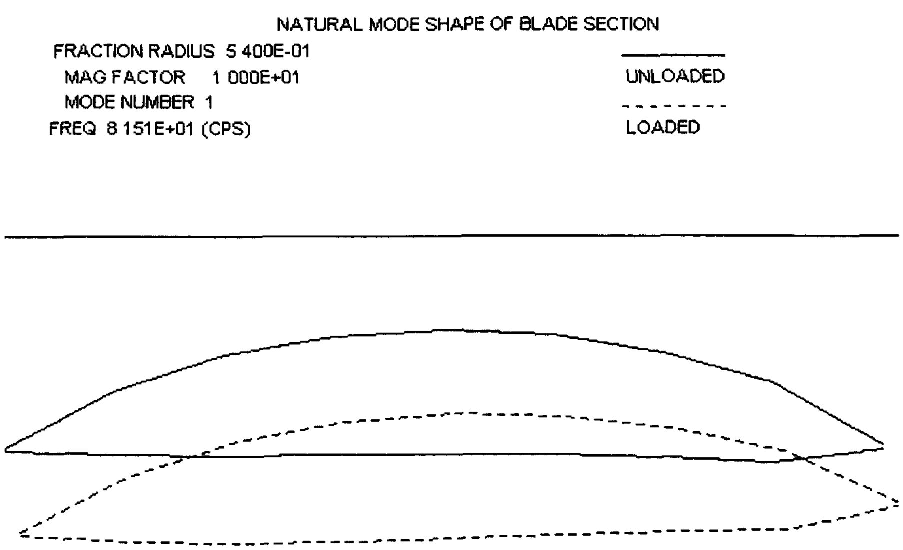


Figure 3.12: Blade Section Mode Shape (Mode 1, at 0.5400 Radial Fraction)

The blade section mode shape for other modes and radial positions can be plotted in a similar manner.

3.4 Blade Only Model of P5363 Propeller Blade

In this section, the generation of finite element models of the *Blade Only* model of the skewed propeller blade (P5363 propeller) is presented. It is assumed that the blade geometry has been generated as described in Chapter 2 and that a PREFIX.T16 file of the blade now exists, where PREFIX is the prefix of the filename for identifying the model. The remaining steps for defining the finite element model include:

(a) selection of model type
(b) generation of finite element grid proportions;
(c) generation of the finite element mesh; and
(d) generation of boundary conditions.

These are presented below.

Model Type

The model type selected is **Blade Only** (see Figure 3.1)

Finite Element Grid Proportions

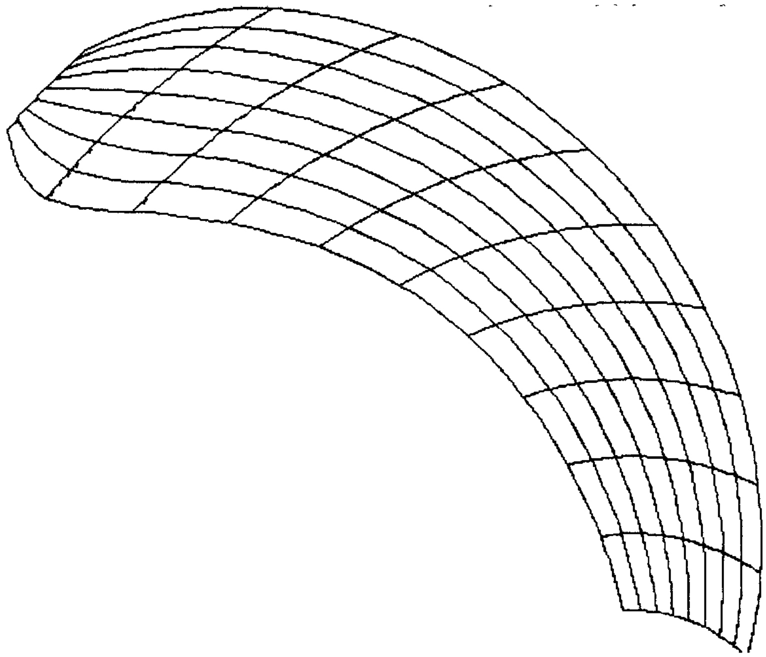
Figure 3.13 shows the grid proportion data entered and Figure 3.14 shows the FE grid proportions obtained.

CREATE AND EDIT FE GRID

PARAMETER	RADIAL DIR.	CHORDWISE DIR.
NO. OF FE	10	8
GRID PROPORTION	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	9.990E-01
GEOM FACTOR	1 000E+00	1 000E+00
OPTIMIZATION	NO OPTIMIZATION	N/A
FRACTION DATA	N/A	N/A

Ok Cancel

Figure 3.13: FE Grid Proportions Data for P5363 Propeller Blade



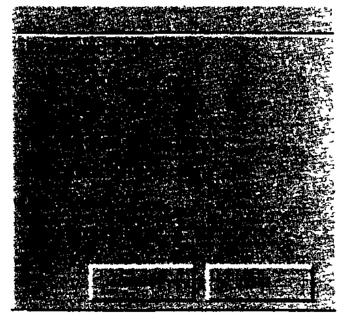


Figure 3.14: FE Grid Proportions for P5363 Propeller Blade

FE Mesh Generation

Figure 3.15 shows the finite element data entered. Figure 3.16 shows a wire frame mesh of model, using the general plotting capability. Figure 3.17 shows the default boundary conditions generated automatically by the program.

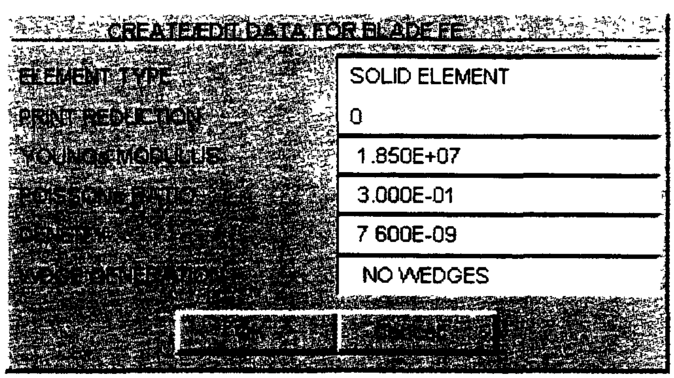


Figure 3.15: Finite Element Mesh Data for P5363 Propeller Blade

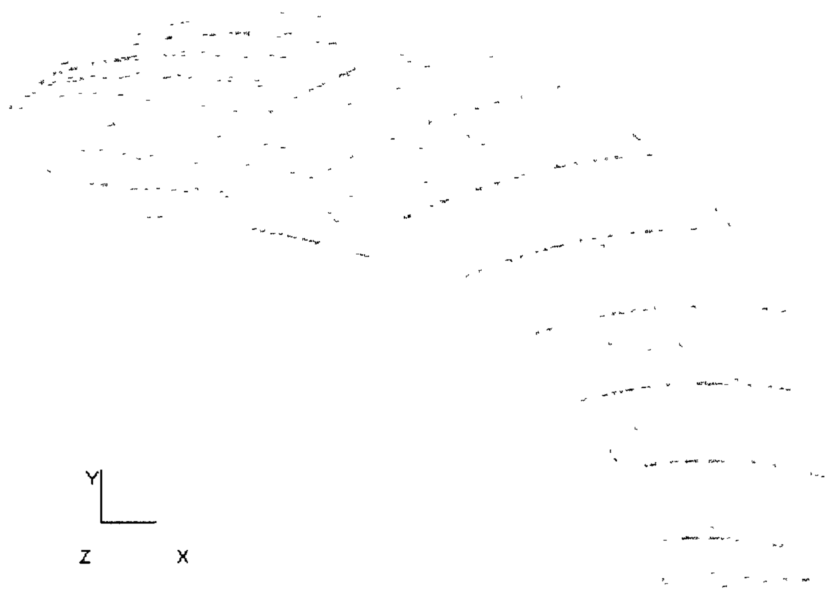


Figure 3.16: Finite Element Model of P5363 Propeller Blade

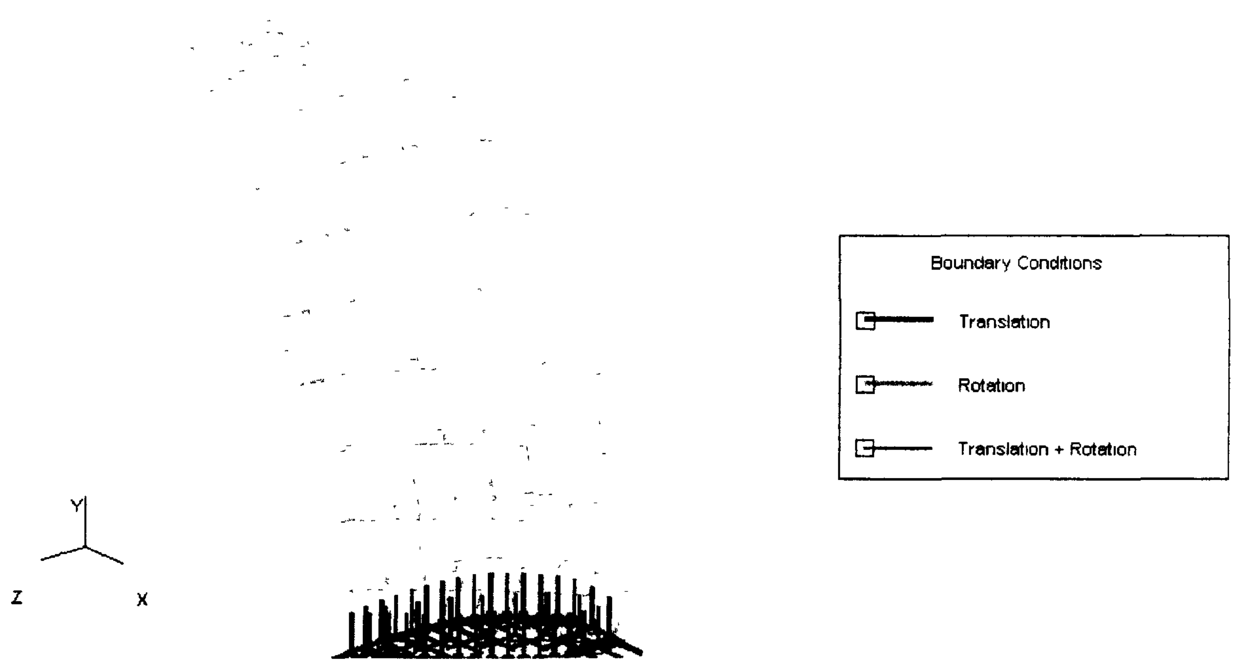


Figure 3.17: Boundary Conditions for P5363 Blade Only Model

3.5 Blade Only Model of P4388 Propeller Blade

In this section, the generation of finite element models of the *Blade Only* model of the highly skewed propeller blade (P4388 propeller) is presented. It is assumed that the blade geometry has been generated as described in Chapter 2 and that a PREFIX.T16 file of the blade now exists, where PREFIX is the prefix of the filename for identifying the model. The remaining steps for defining the finite element model include:

(e) selection of model type
(f) generation of finite element grid proportions;
(g) generation of the finite element mesh; and
(h) generation of boundary conditions.

These are presented below.

Model Type

The model type selected is **Blade Only** (see Figure 3.1)

Finite Element Grid Proportions

Figure 3.18 shows the grid proportion data entered and Figure 3.19 shows the FE grid proportions obtained.

CREATE AND EDIT FE GRID

PARAMETER	RADIAL DIR.	CHORDWISE DIR.
NO. OF FE	20	10
GRID PROPORTION	BIASED TO TIP	BIASED LE/TE
NO. OF POSITIONS	N/A	1
1ST FRACTION	2 000E-01	1 000E-03
LAST FRACTION	9 800E-01	9.990E-01
[illegible]	1 000E+00	1.000E+00
[illegible]	NO OPTIMIZATION	N/A
[illegible]	N/A	N/A

Figure 3.18: FE Grid Proportions Data for P4388 Propeller Blade

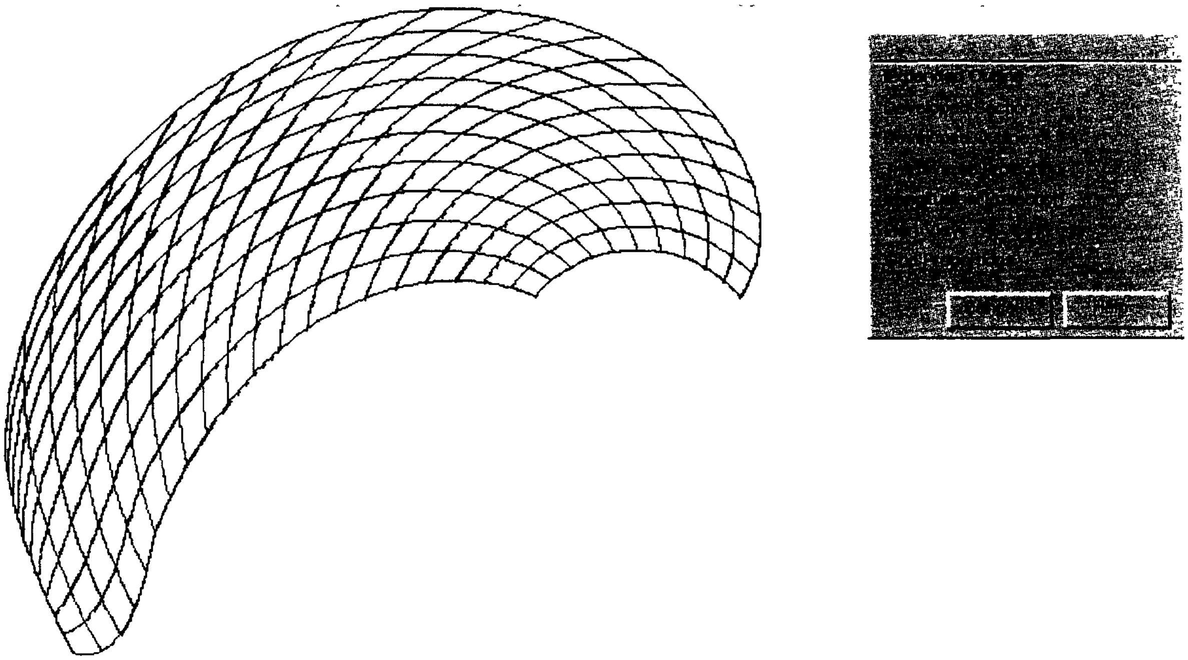


Figure 3.19: FE Grid Proportions for P4388 Propeller Blade

FE Mesh Generation

Figure 3.20 shows the finite element data entered. Figure 3.21 shows a fill plot of the model, using the general plotting capability. Figure 3.22 shows the default boundary conditions generated automatically by the program.

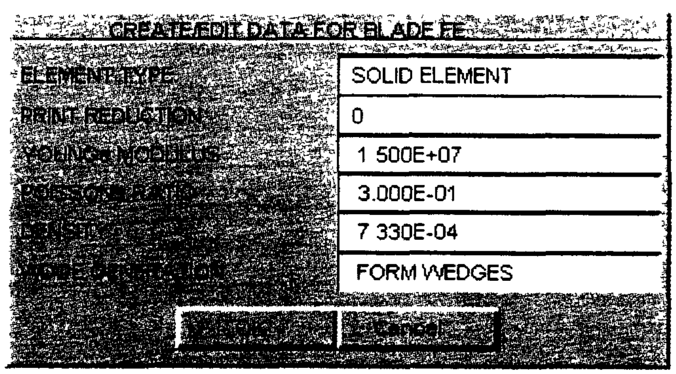


Figure 3.20: Finite Element Mesh Data for P4388 Propeller Blade

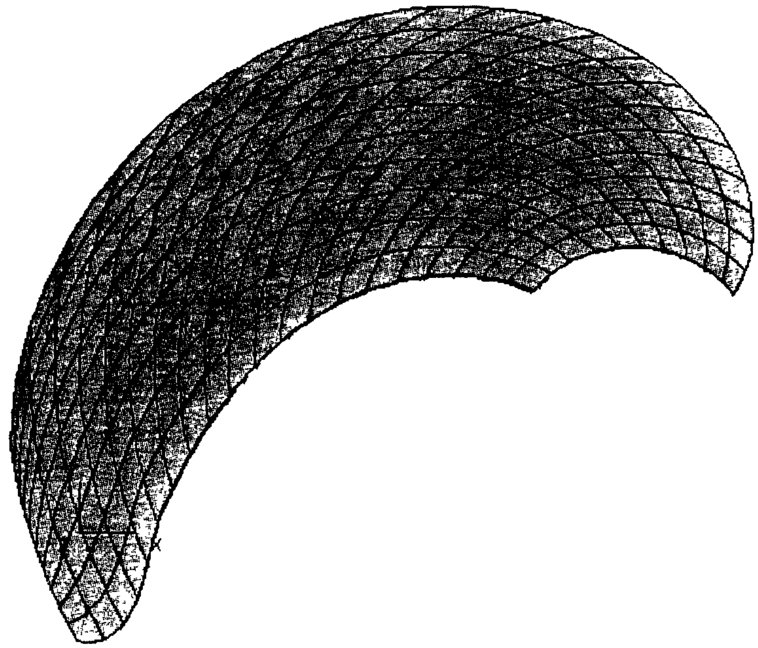


Figure 3.21: Finite Element Model of P4388 Propeller Blade

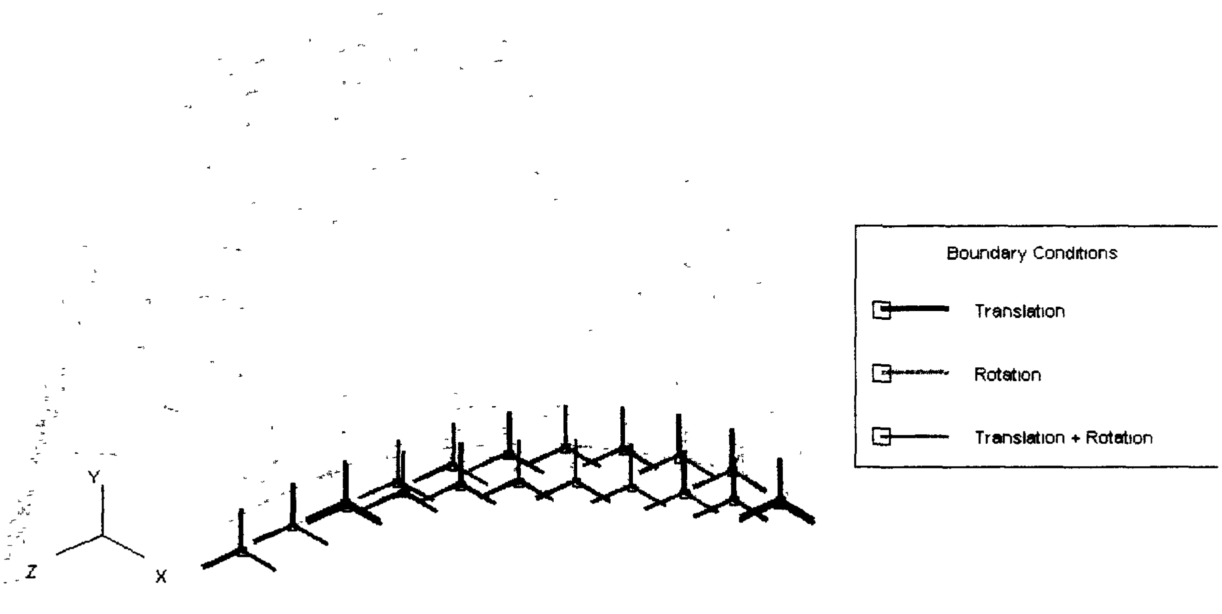


Figure 3.22: Boundary Conditions for P4388 Blade Only Model

4. BLADE ONLY MODEL FOR VIBRATION ANALYSIS IN WATER

4.1 Introduction

In this chapter, we shall demonstrate the how to generate a propeller blade finite element model, including the fluid finite element model, perform a natural frequency analysis of the model that includes the fluid the added mass; and to view the results. The narrow blade (NRC45) propeller blade shall be used for this demonstration. A natural frequency analysis of this blade in air was considered in the Chapter 3. Thus the blade finite element is available at this point, and the user can refer to Chapter 3 for a description of the steps necessary to generate the finite element model. The steps involved in this analysis are:

(i) To select the model type (**Blade Only**)
(ii) To generate the blade geometry
(iii) To generate the finite element grid proportions
(iv) To generate the blade finite element model
(v) To generate the boundary conditions
(vi) To generate the fluid finite element model
(vii) To perform the finite element analysis using the VAST program; and
(viii) To post-process and view the results

With the blade finite element model already defined, only the last three steps are required for the present analysis. These steps are described in sections 4.2 to 4.4.

There are two types of fluid models that can be generated. These include the full 3D-fluid finite element model and the surface panel fluid model. While reasonable surface panel fluid models can be generated for any propeller blade, the full 3-D fluid model option has to be used with care especially for highly skewed propeller blades to avoid the generation of distorted fluid elements. To illustrate the applicability of the PVAST algorithm for various blade configurations, fluid finite element models are also generated for two other blades – one a skewed blade (P5363 propeller), and the other a highly skewed blade (P4388 propeller) – in Section 4.5

4.2 Fluid Finite Element Model Generation

There are two types of fluid finite element models that can be generated. The first is a full 3-D fluid finite element model, and the other is a surface panel model. These are discussed below.

4.2.1 Full 3-D Fluid Finite Element Model

4.2.1.1 Generating the Fluid FE Model

The 3-D fluid finite element model can be generated according to the following steps:

Click the **Create/Modify** on the main menu and select the **Fluid FE** option on the **CREATE/MODIFY** submenu (Figure 3.2).

Figure 4.1(a) shows the edit table for creating/editing fluid model data. In the table, the type of fluid model is set to **Finite Element Model**, and the original finite element grid of the blade is used as the fluid-structure interface grid. Default values are entered for all the other required data.

Click the **Create/Edit** box to enter the fluid layer depth data, as shown in Figure 4.1(b). These are provided as fractions of the full blade radius.

Click **OK** when finished.

At this point, a pick menu (Figure 4.2) is presented for selecting the kind of fluid model to plot. Click **Thin Fluid Model** to generate the thin fluid model. This is a model of the full 3-D fluid model, with the fluid layer thickness reduced to zero. This is used to first check the outline of the full 3-D model, before it is actually generated. Once the selection is made, the program generates the thin fluid model and provides the message *Finished Generating Fluid FE Model* at the bottom left corner of the screen.

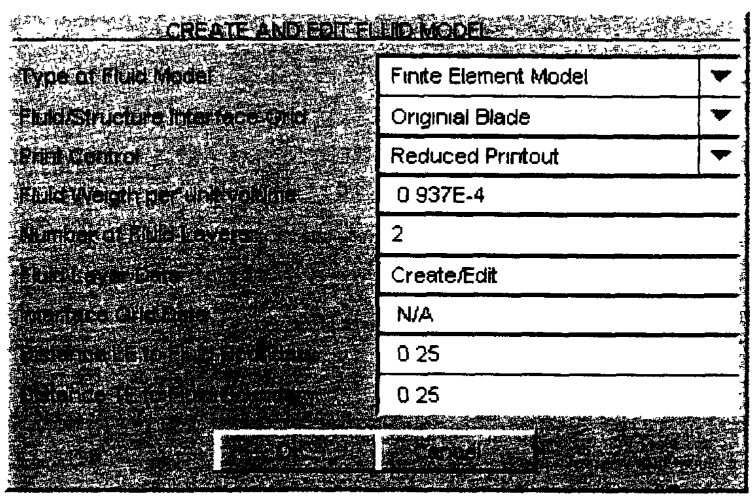


Figure 4.1(a): Blade Geometry Menu

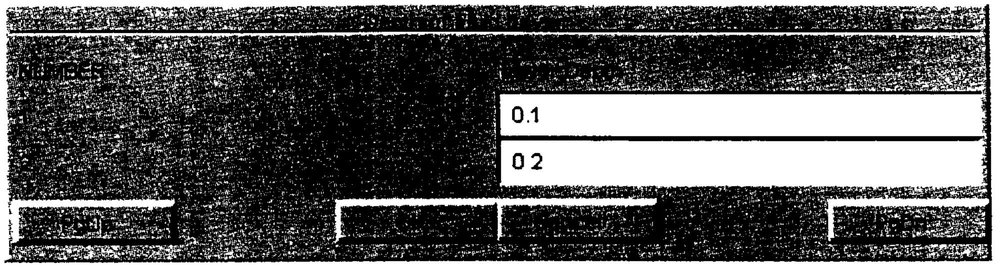


Figure 4.1(b): Entering Fluid Layer Depth Data

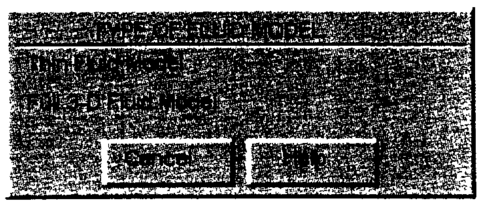


Figure 4.2: Selecting the Kind of Fluid FE Model

After generating the thin fluid model, the model can be viewed as described in Section 4.2.1.2. If the thin blade outline is satisfactory, proceed to generate the full 3-D fluid finite element model by selecting the **Full 3-D Fluid Model** option on Figure 4.2. Again, the program generates the fluid model and provides the message *Finished Generating Fluid FE Model* when finished. The fluid model can now be viewed as described below.

4.2.1.2 Displaying the Fluid FE Model

From the PVAST main menu, click **Display.**

4.2.1.3 For Special Plotting Option

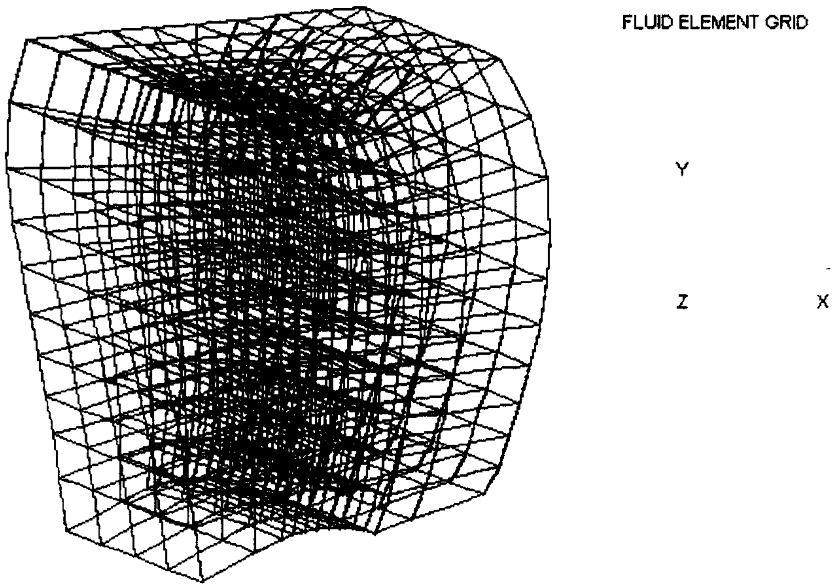
Click **Special Plots** on the **MODEL DISPLAY** submenu.
Click **Fluid Model** on the **SPECIAL PLOTS** submenu

The fluid FE model will then be displayed as shown in Figure 4.3(a).

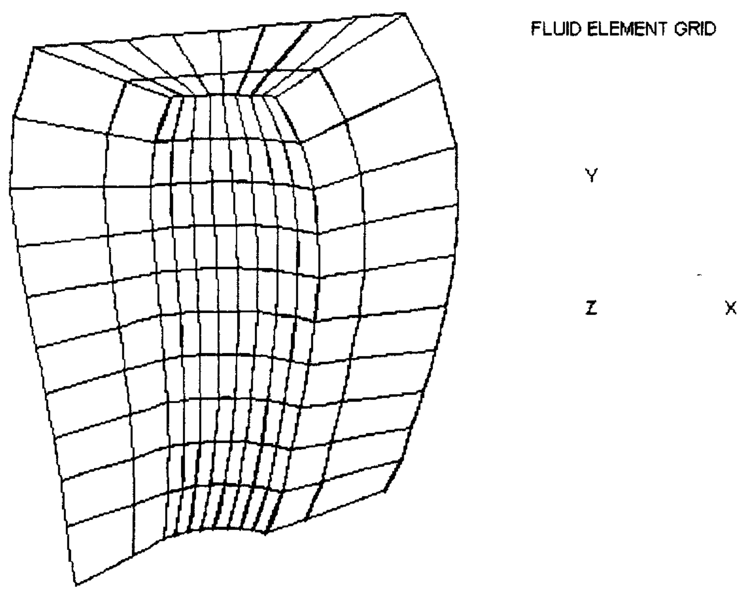
The thin fluid model (if available) can also be displayed by clicking the options button on this screen (not shown in figure), and selecting the **Thin Model** option. The thin fluid model is shown in Figure 4.3(b).

For general plotting option
Click **Fluid Model** on the **MODEL DISPLAY** submenu.

The fluid model will be generated using the MGDSA general purpose plotting capabilities, as shown in Figure 4.4.



(a)



(b)

Figure 4.3: Fluid Finite Element Models Using Special Plotting Option: (a) Full 3-D Model, (b) Thin Fluid Model

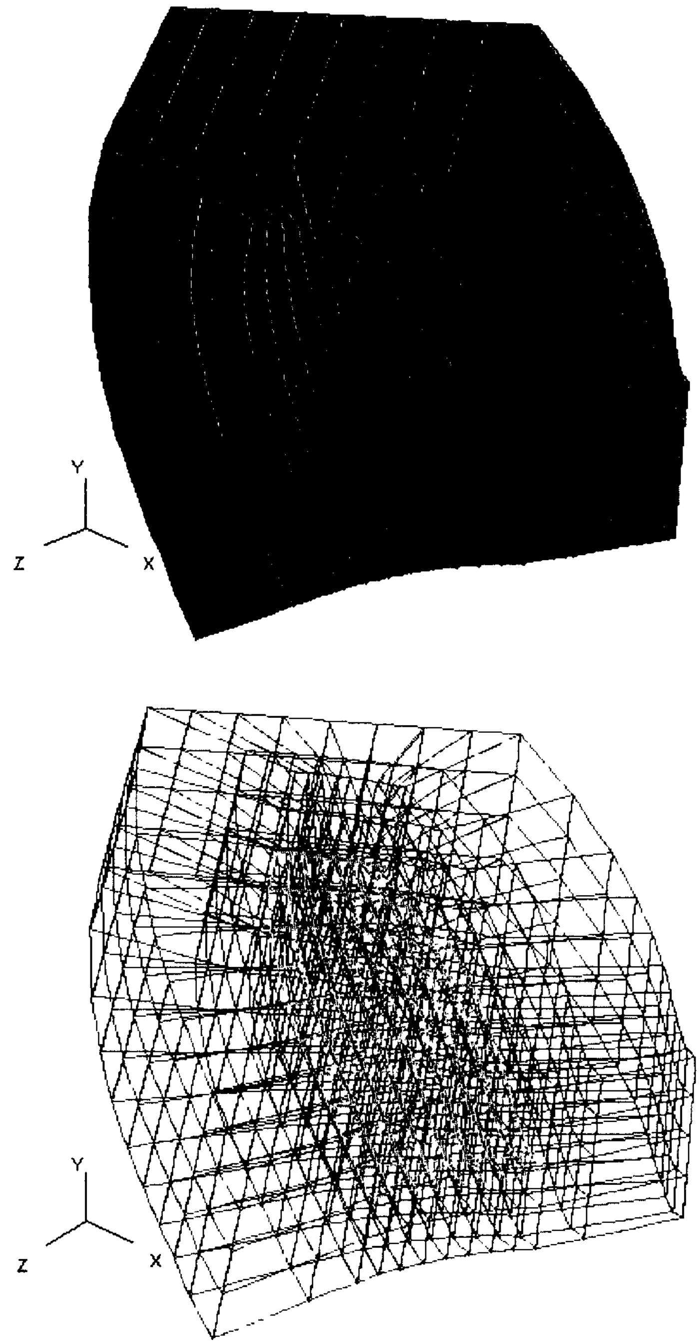


Figure 4.4: Fluid Finite Element Models Using General Plotting Option: (a) Fill Plot, (b) Wire Mesh

4.2.2 Surface Panel Fluid Model

4.2.2.1 Generating the Surface Panel Fluid Model

The surface panel fluid model can be generated according to the following steps:

Click the **Create/Modify** on the main menu and select the **Fluid FE** option on the **CREATE/MODIFY** submenu (Figure 3.2). Click **Type** of Fluid Model and select **Surface Panel Model**.

Figure 4.5 shows the edit table for creating/editing fluid model data. In the table, the type of fluid model is set to **Surface Panel Model**, and free surface effects are ignored in this model. Default values are entered for all the other required data.

Click **OK** when finished.

At this point, the program generates the surface panel fluid model and provides the message *Finished Generating Fluid FE Model* at the bottom left corner of the screen.

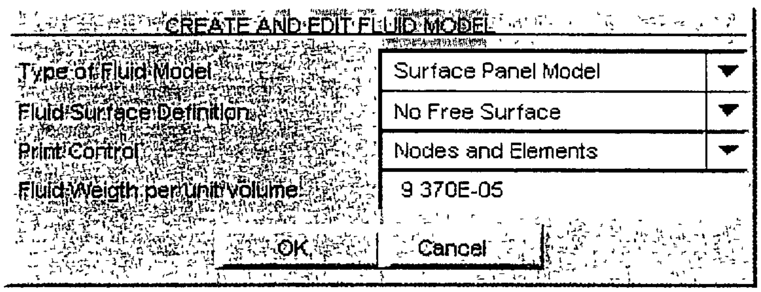


Figure 4.5: Edit Table for Creating/Editing the Surface Panel Fluid Model

4.2.2.2 Displaying the Fluid FE Model

From the PVAST main menu, click **Display.**

Click **Fluid Model** on the **MODEL DISPLAY** submenu.

The fluid surface panel model will then be displayed as shown in Figure 4.6.

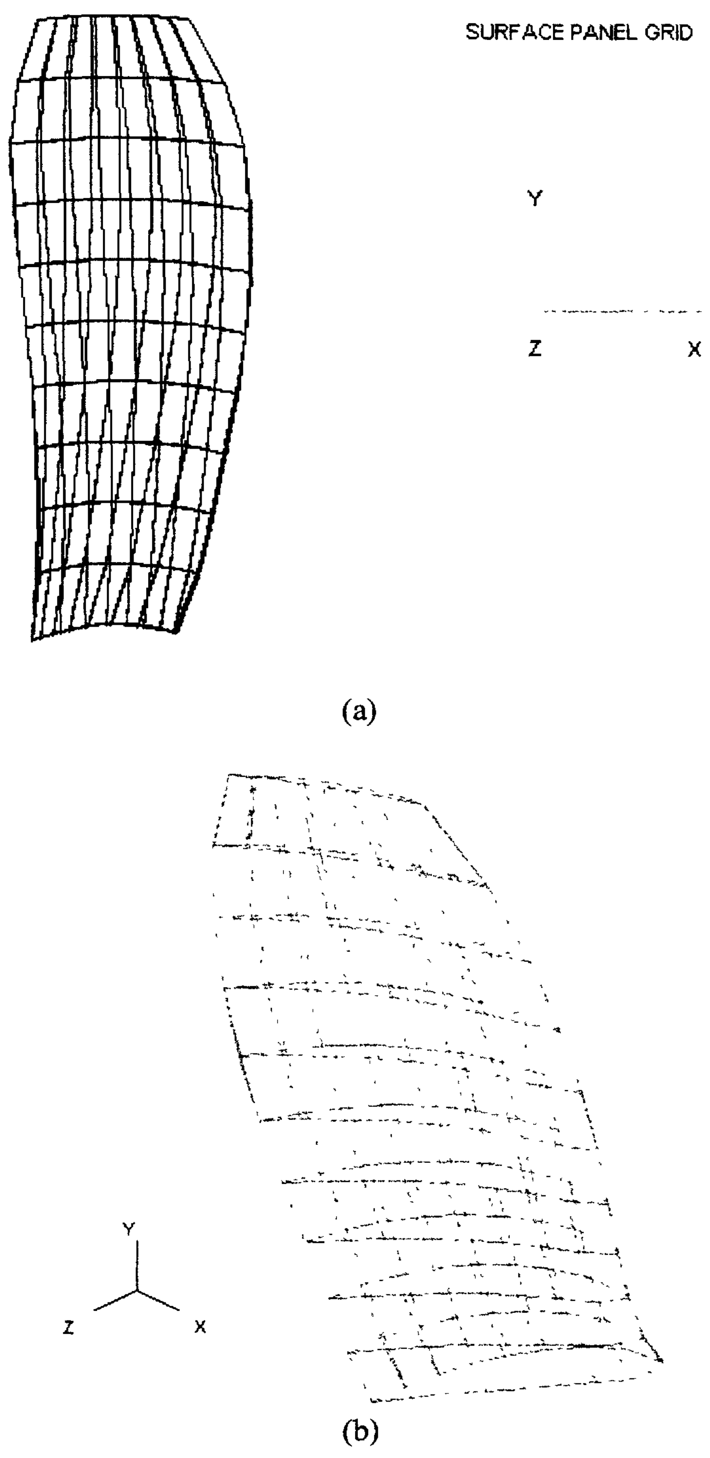


Figure 4.6: Fluid Surface Panel Model of NRC45 Blade: (a) Special Plot Option, (b) General Plot Option

4.3 VAST Finite Element Analysis

This step involves the performance of the finite element analysis using the VAST finite element code. This is accomplished by the following processes:

4.3.1 Generate VAST Input File

This step involves generating the input file for a VAST finite element analysis.

Click the **Create/Modify** on the main menu and select the **VAST Input File** option on the **CREATE/MODIFY** submenu (Figure 3.2).

Select **Natural Frequency** analysis, and include **Fluid Mass** as shown in Figure 4.7. Click **OK.**

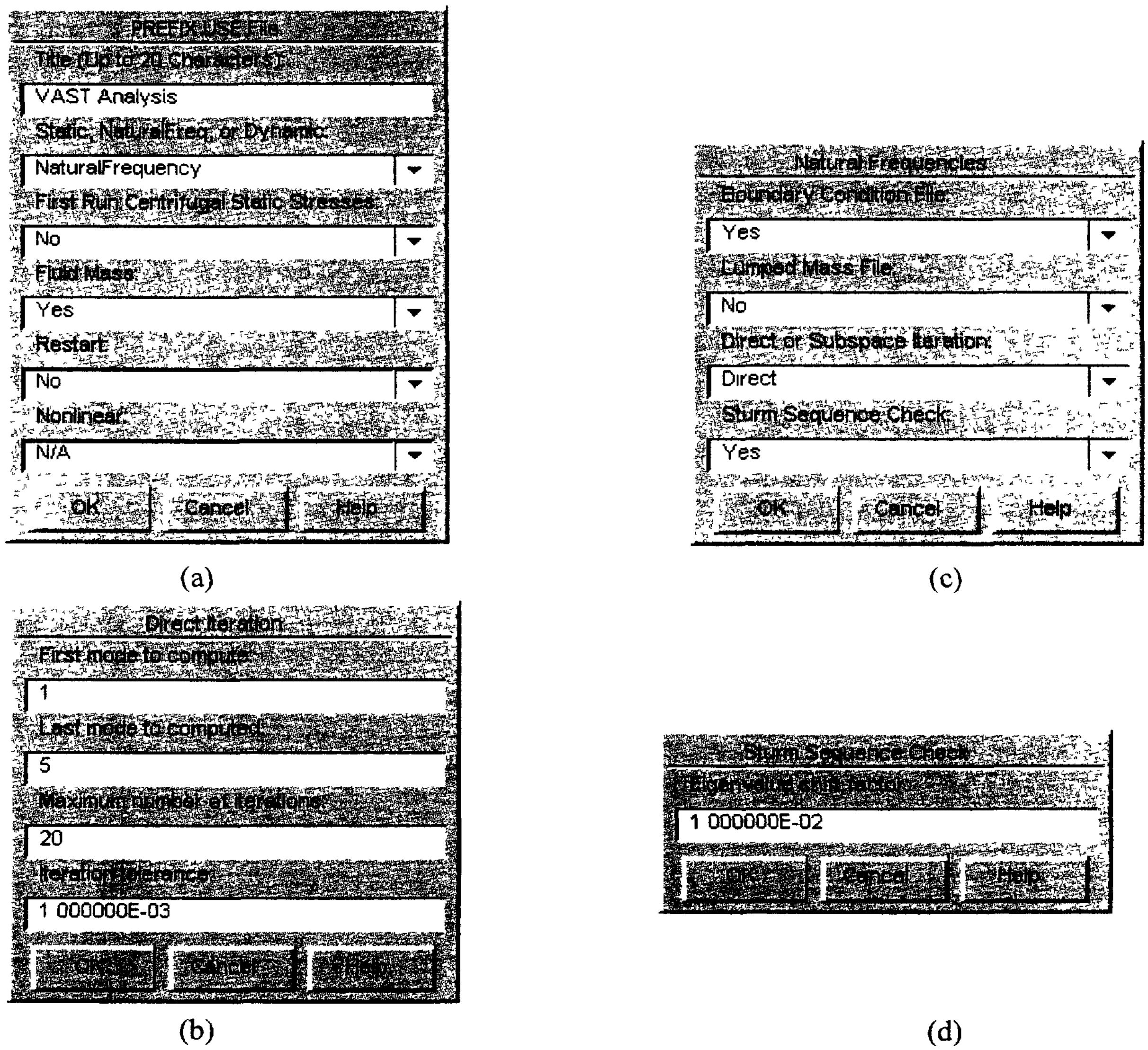


Figure 4.7: Generating VAST Input File for Natural Frequency Analysis in Water

4.3.2 Natural Frequency Analysis

The procedure is exactly the same as described in Section 3.3.2.

Click **Analysis** on the main menu. Than Click **Start VAST Analysis** on the **Analysis** submenu as shown in Figure 4.8.

Figure 4.8: Analysis Submenu

The VAST analysis status, shown in Figure 4.9 will now be displayed. Click **Start** to start the VAST natural frequency analysis. The progress of the analysis will be shown in the dialog box, and a message will be provided when the analysis is completed. Click **OK** and return to the PVAST main menu.

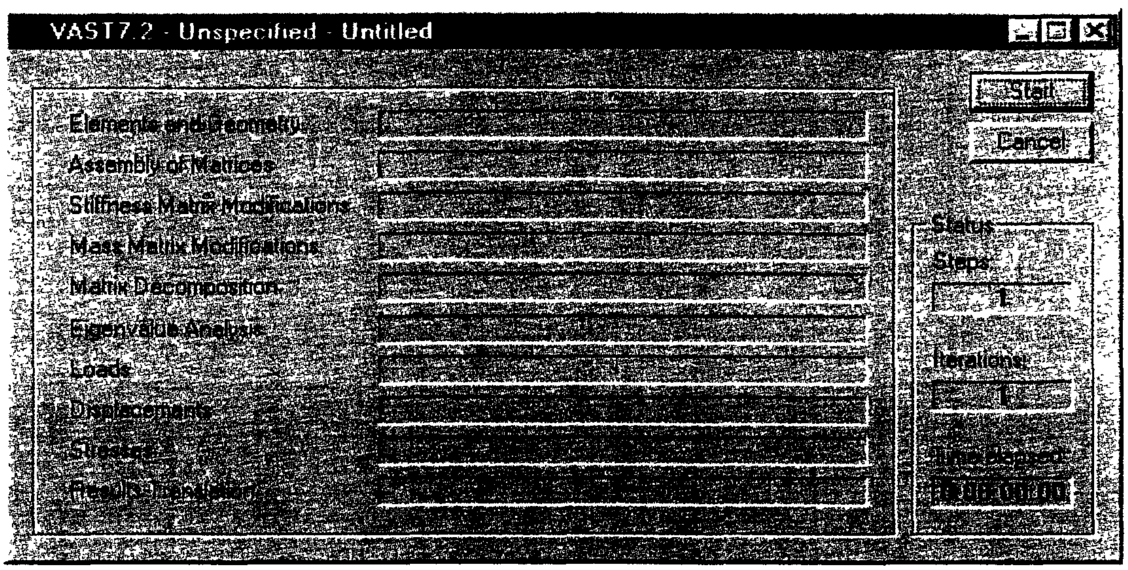
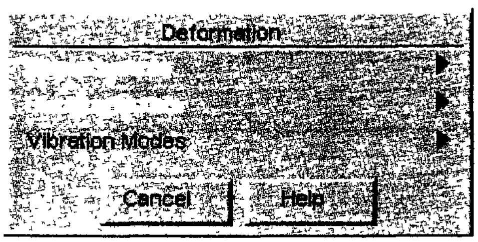


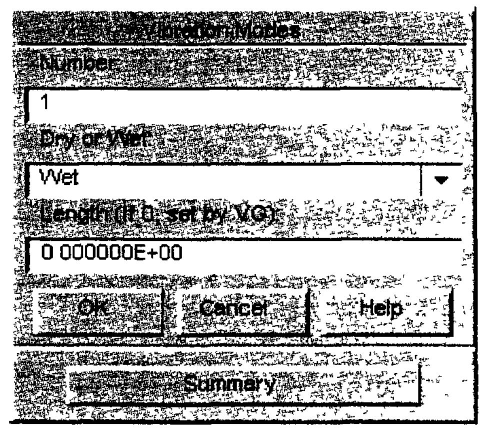
Figure 4.9: VAST Analysis Status Dialog Box

4.4 Post-Processing of Results

The plotting of results is similar to that described in Section 3.4. However, in this case mode shapes of *dry* and *wet* modes are available for plotting. The dry modes are plotted exactly as described in Section 3.4, and since the results are similar to those in obtained in Section 3.4, they are not repeated here. For wet modes, only the general plotting capability is used as the special plotting capability is not available for this case. To plot the wet modes using the general plotting capability, the following steps should be followed:

Click **Results** on the main menu (Figure 3.12 (a) appears).
Click **Deformation** on the **Results** submenu (Figure 4.10(a) appears).
Click **Vibration Modes,** (Figure 4.10 (b) appears).
Enter the desired mode number. Select **Wet** mode and enter a scale factor **(Length)** as shown. The corresponding mode shape (using wire mesh plotting) is shown in Figure 4.11.





(a) **(b)**

Figure 4.10: Menus for General Plots (Wet Mode Results)

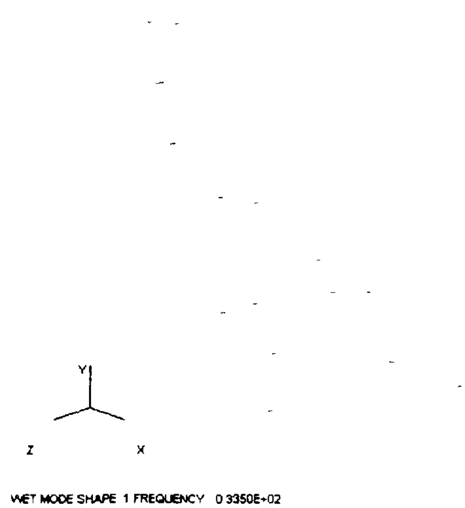
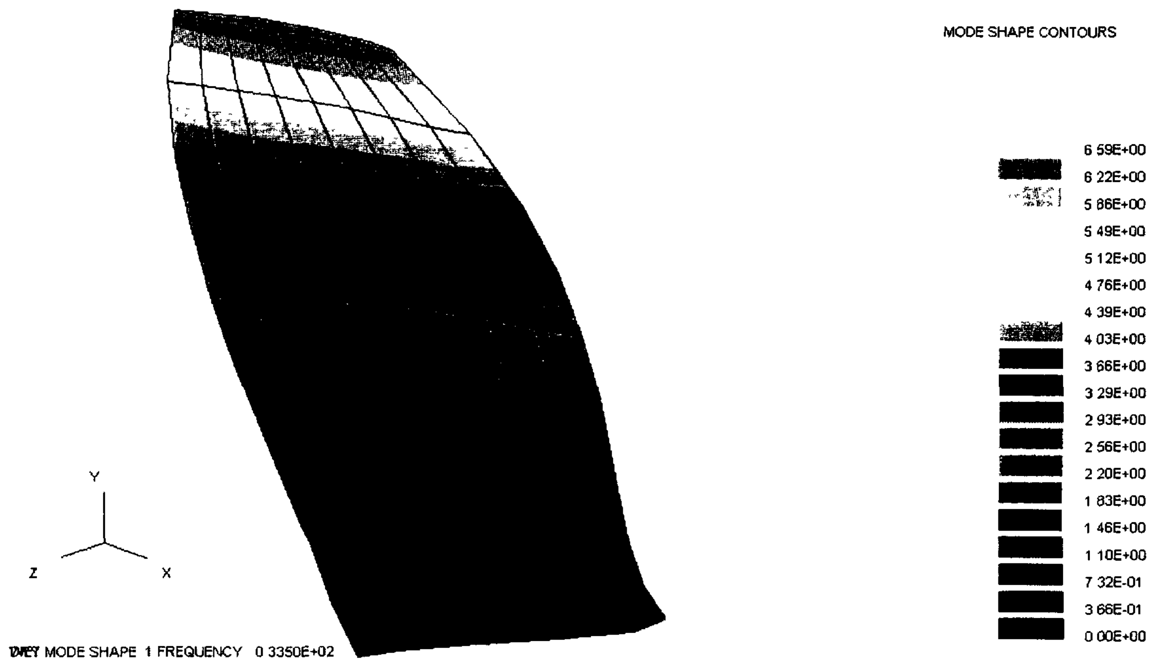


Figure 4. 11: Wire Mesh Plot of Mode 1 of NRC45 Propeller Blade (Wet Mode)

Plotting Contours of Mode Shapes

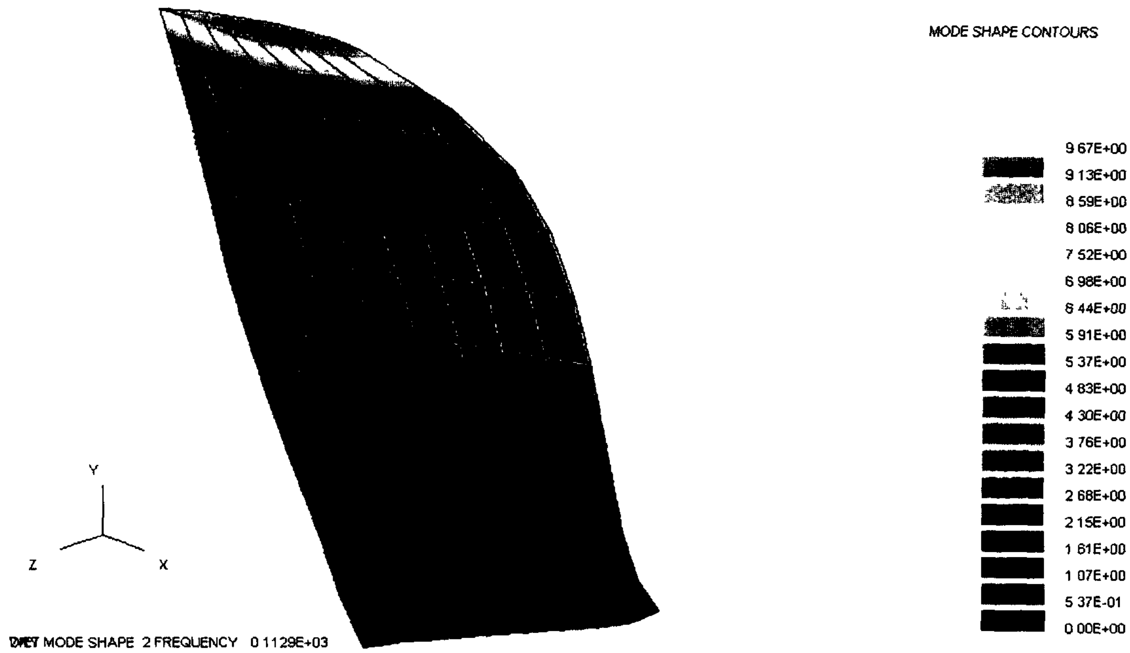
Plotting wet mode shape contours involves the following steps:
Click **Options** on the main menu
Click **Plotting** on the **Options** submenu
Click **Contour** on the **Plotting** submenu
Click **Deformation** on the **Contour** submenu
Select **Fringe** plotting

Figures 4.12 (a)-(c) show the contours of wet modes 1-3, respectively.

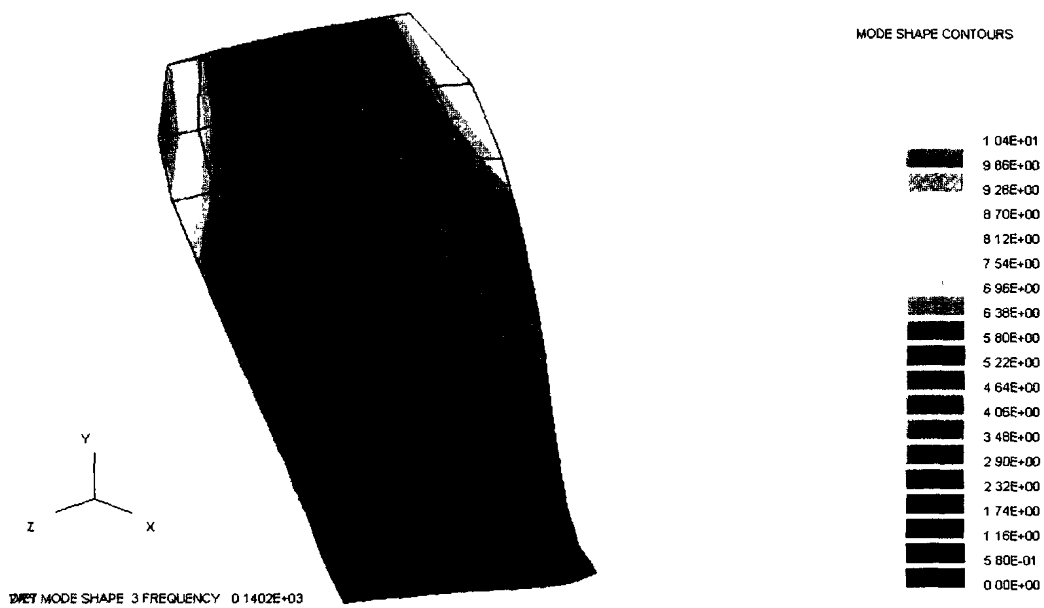


(a)

Figure 4.12: Contours of Mode Shapes of NRC45 propeller in Water: (a) Mode 1



(b)



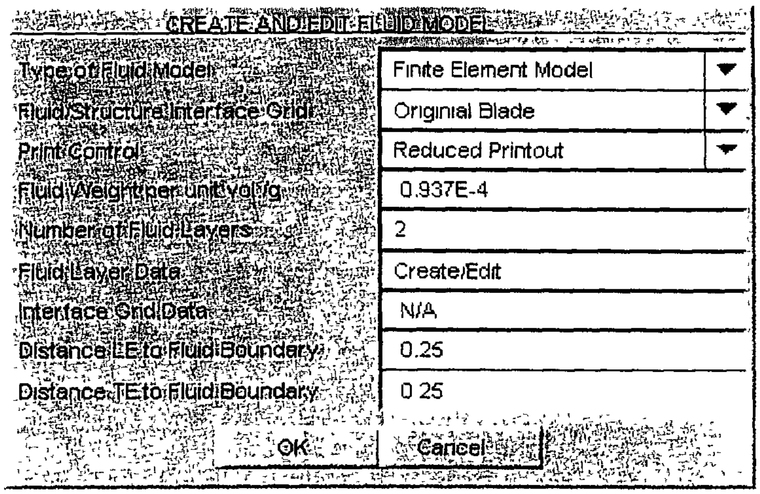
(c)

Figure 4.12: Contours of Mode Shapes of NRC45 propeller in Water: (a) Mode 1; (b) Mode 2; and (c) Mode 3

4.5 Fluid Models for P5363 Propeller Blade

In this section, the generation of fluid finite element models of the *Blade Only* model of the skewed propeller blade (P5363 propeller) is presented. It is assumed that the blade geometry and the finite element model of the blade have been generated as described in Chapters 2 and 3. Only the fluid finite element model generation step is considered here.

Figure 4.13 shows the data entered for the fluid model generation.



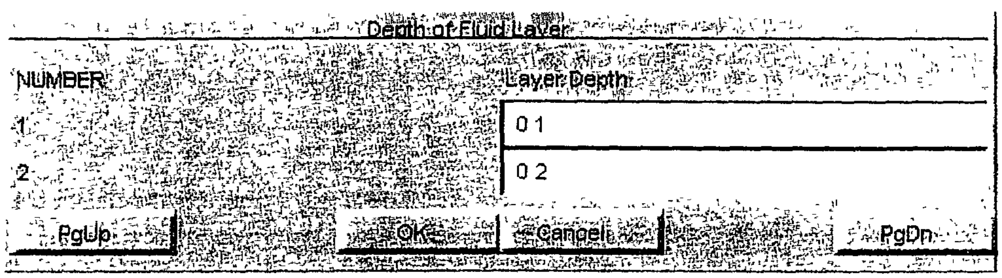


Figure 4.13: Fluid Model Data for P5363 Propeller Blade

Figure 4.14 shows the thin fluid finite element model using the special plotting option. Figure 4.15(a) shows a wire mesh of the 3-D fluid finite element model superimposed on the blade model, and Figure 4.15(b) shows a fill plot of the fluid model.

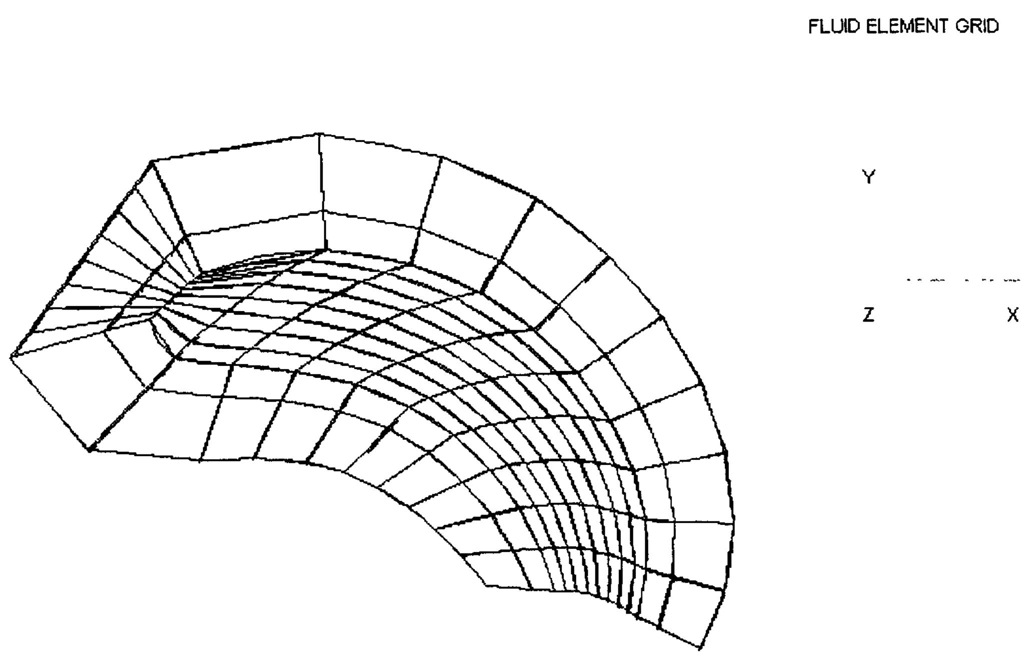
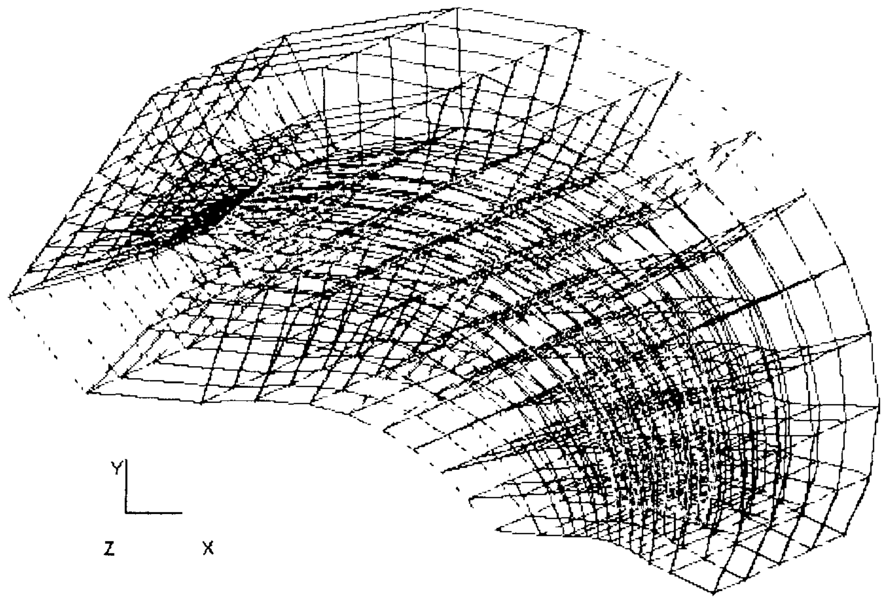
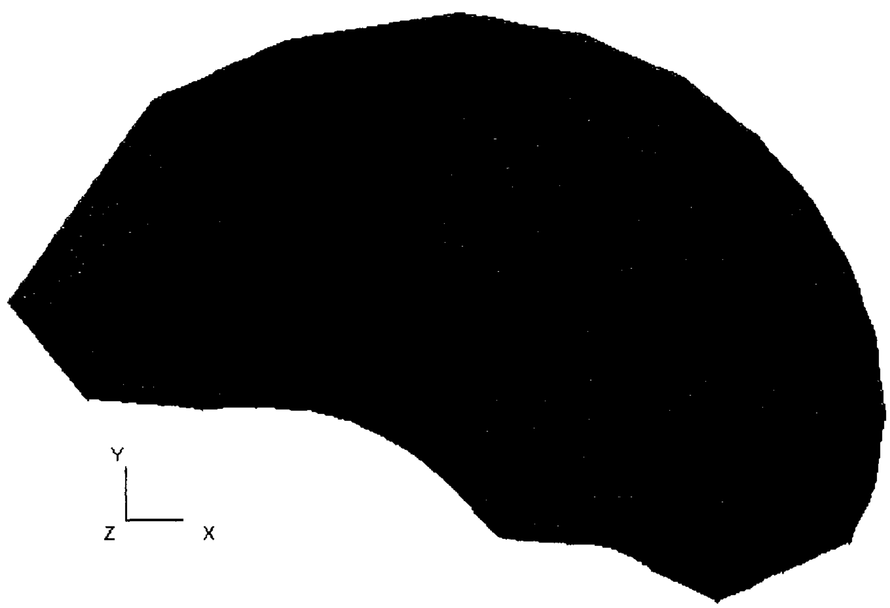


Figure 4.14: Thin Fluid Model of P5363 Propeller Blade



(a)



(b)

Figure 4.15: Full 3-D Finite Element Meshes of P5363 Propeller Blade; (a) Wire mesh of fluid superimposed on Blade; and (b) Fill Plot of Fluid Model

4.6 Fluid Models for P4388 Propeller Blade

In this section, the generation of fluid finite element models of the *Blade Only* model of the highly skewed propeller blade (P4388 propeller) is presented. It is assumed that the blade geometry and the finite element model of the blade have been generated as described in Chapters 2 and 3. Only the fluid finite element model generation step is considered here.

Figure 4.16 shows the data entered for the fluid model generation.

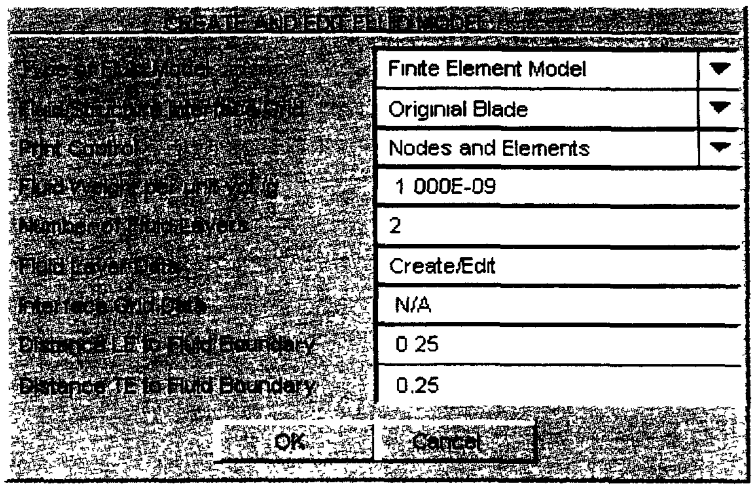


Figure 4.16: Fluid FE Data for P4388 Propeller

Figure 4.17 shows the thin fluid finite element model using the special plotting option. Figure 4.18(a) shows a wire mesh of the 3-D fluid finite element model superimposed on the blade model, and Figure 4.18(b) shows a fill plot of the fluid model.

Due to the highly skewed nature of this blade, care had to be exercised in generating the fluid model. Some of the measures taken to avoid problems are highlighted below.

- The distance from the blade trailing edge to the fluid boundary (**Distance TE to Fluid Boundary**) should not be too largen otherwise, the fluid elements may cross over each other.
- A fairly fine mesh is used to have better distribution of the element, especially toward the tip. In this example a 20x10 blade FE mesh is used. An earlier model involving a 10x10 grid did not produce well shaped fluid elements near the tip.

Similar corrective measures might be required for other complex shaped propeller blade geometries.

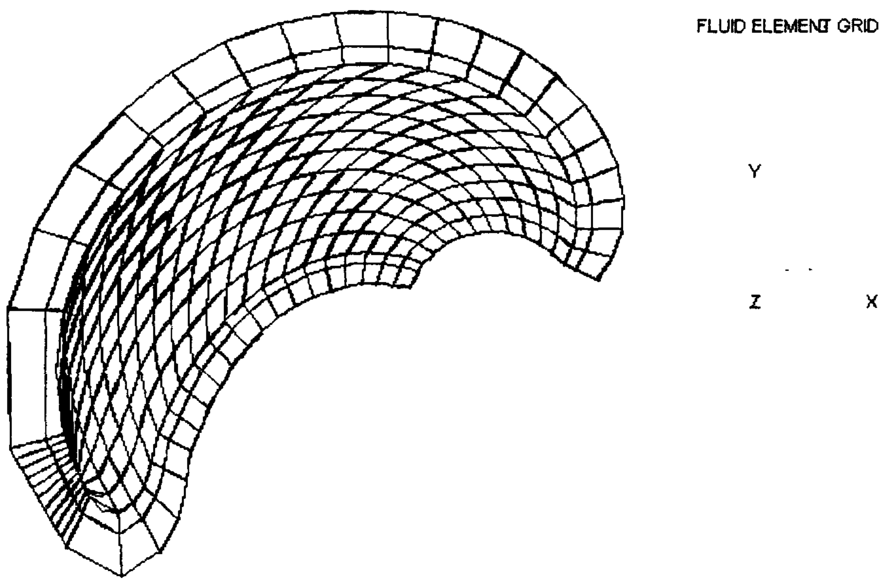
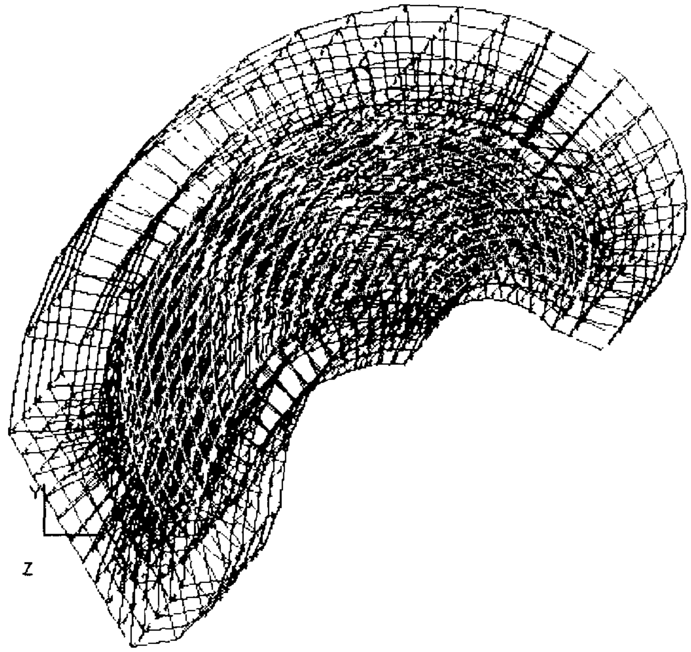
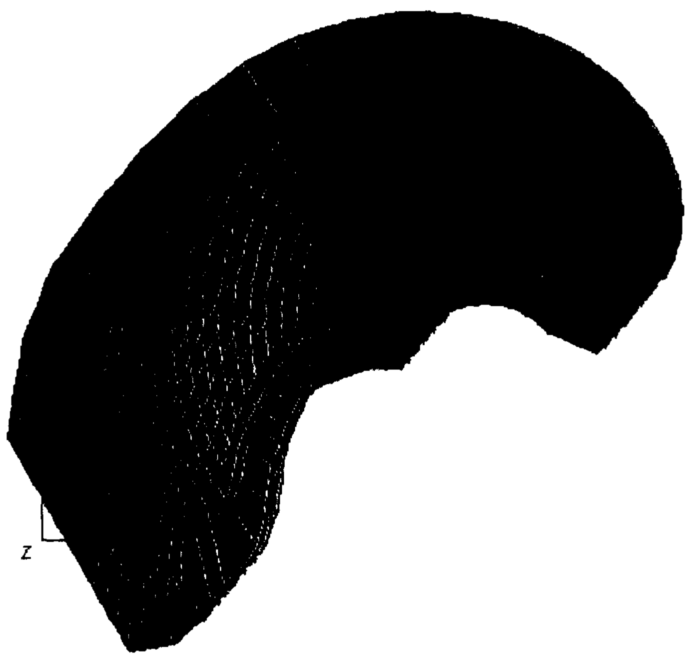


Figure 4.17: Thin Fluid Model of P4388 Propeller Blade



(a)



(b)

Figure 4.18: Full 3-D Finite Element Meshes of P4388 Propeller Blade; (a) Wire mesh of fluid superimposed on Blade; and (b) Fill Plot of Fluid Model

A surface panel fluid model was also generated for this blade. Figure 4.19 shows the data entered to generate the surface panel model. Figure 4.20 shows the surface panel model using the general plotting option.

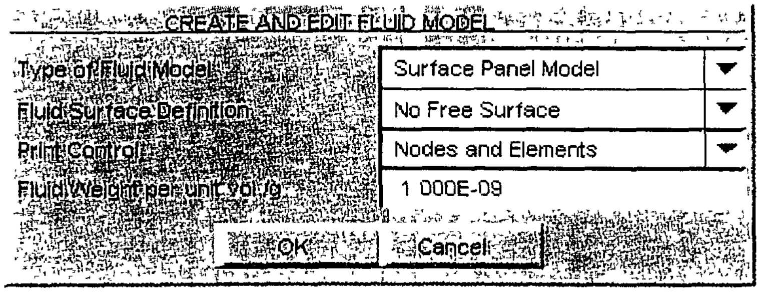


Figure 4.19: Surface Panel Data for P4388 Propeller Blade

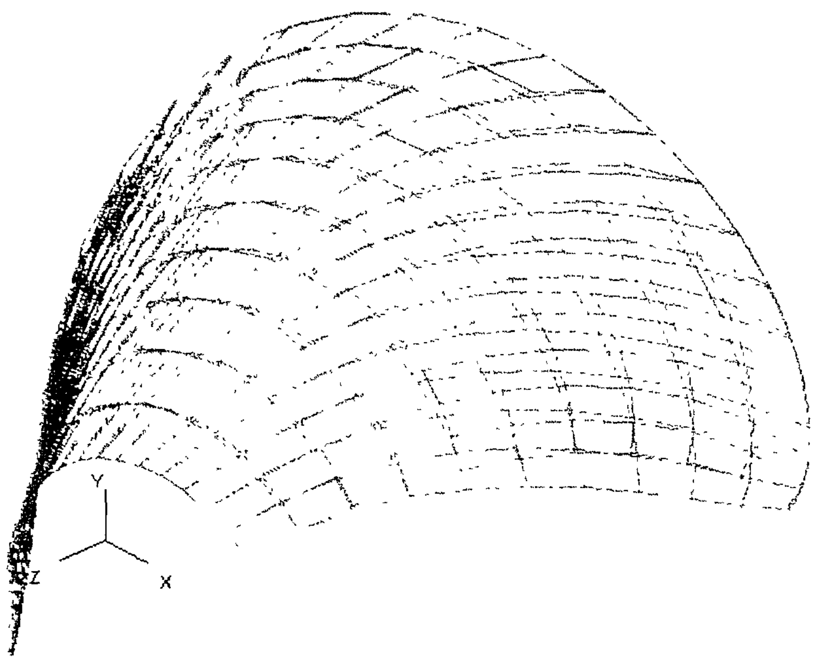


Figure 4.20: Surface Panel Fluid Model for P4388 Propeller Blade

5. BLADE ONLY MODEL FOR STRESS ANALYSIS

5.1 Introduction

In this chapter, we shall demonstrate how to generate a propeller blade finite element model, perform a static stress analysis of the model; and to view the results. It is assumed that the blade geometry data has been entered and geometry defined, as described in Chapter 2. Thus, a PREFIX.T16 file is now available in the working directory, where, "PREFIX" is the alphanumeric prefix of the file name. It is also assumed that the hydrodynamic loads are available on the PREFIX.LIN file. The skewed blade (P5363) propeller blade shall be used for this demonstration. With the blade geometry already defined, the remaining steps are:

(i) To generate the finite element model. This includes the selection of the model type, generation of finite element grid proportions; adjusting peak pressure loads; defining nodal pressures and drag pressures; generation of the finite element mesh; generation of loads on the finite element mesh; and generation of boundary conditions.
(ii) To perform the finite element analysis using the VAST program; and
(iii) To post process and view the results

These steps are described in the following sections.

5.2 Finite Element Model Generation

5.2.1 Definition of Model Type

Click **Model** on the main menu and ensure that the **Blade Only** model is highlighted, as shown in Figure 3.1.

5.2.2 Finite Element Grid Proportions

The finite element grid proportions for the P5363 propeller blade were previously defined in Section 3.5. The same grid proportions are used here and the data entered are presented in Figures 3.18 and 3.19. Refer to Section 3.5.

5.2.3 Adjust Peaks of Pressure Loads

The blade pressure load is assumed to be available on a file called P5363.LIN. The contents of the file are shown in Appendix B. The purpose of this step is to adjust the applied load to reduce pressure spikes along the leading and trailing edges from which loading errors could develop. The adjustment is carried out such that the smoothed chordwise pressure distributions produce the same thrust and spindle thrust as the original pressure distribution. The procedure is as follows:

Click the **Create/Modify** on the main menu and select the **Adjust Peaks of Pressure Loads** option on the **CREATE/MODIFY** submenu (Figure 3.2).

The **Adjust Peaks of Pressure Loads** submenu appears as shown in Figure 5.1. The options selected are as shown in the figure.

Click **OK** when finished.

In this demonstration, both the leading edge (LE) and trailing edge (TE) loads are adjusted with no limits, as shown in Figure 5.1.

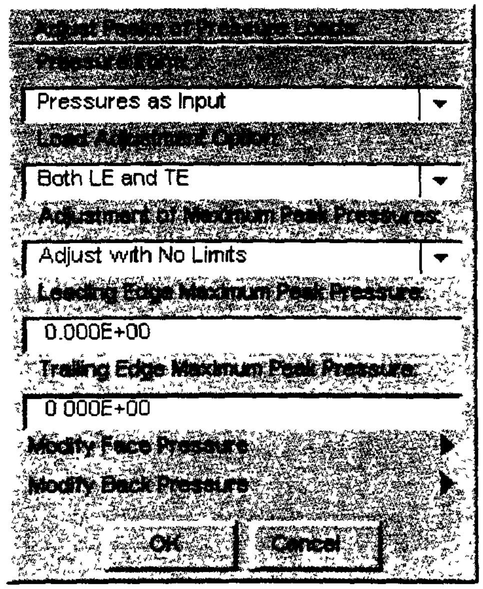


Figure 5.1: Submenu for Adjusting Peaks of Pressure Loads

To display the loads:

Click **Display** on the main menu
Click **Special Plots**
Click **Adjusted Chordwise Pressure**

The pressure loads are displayed as shown in Figures 5.2 to 5.4. Figure 5.2 shows a graph of the pressure load versus fraction chord. This **Load Graph**. Similarly, Figure 5.3 shows an expanded plot showing the pressure superimposed on the expanded outline of the blade.

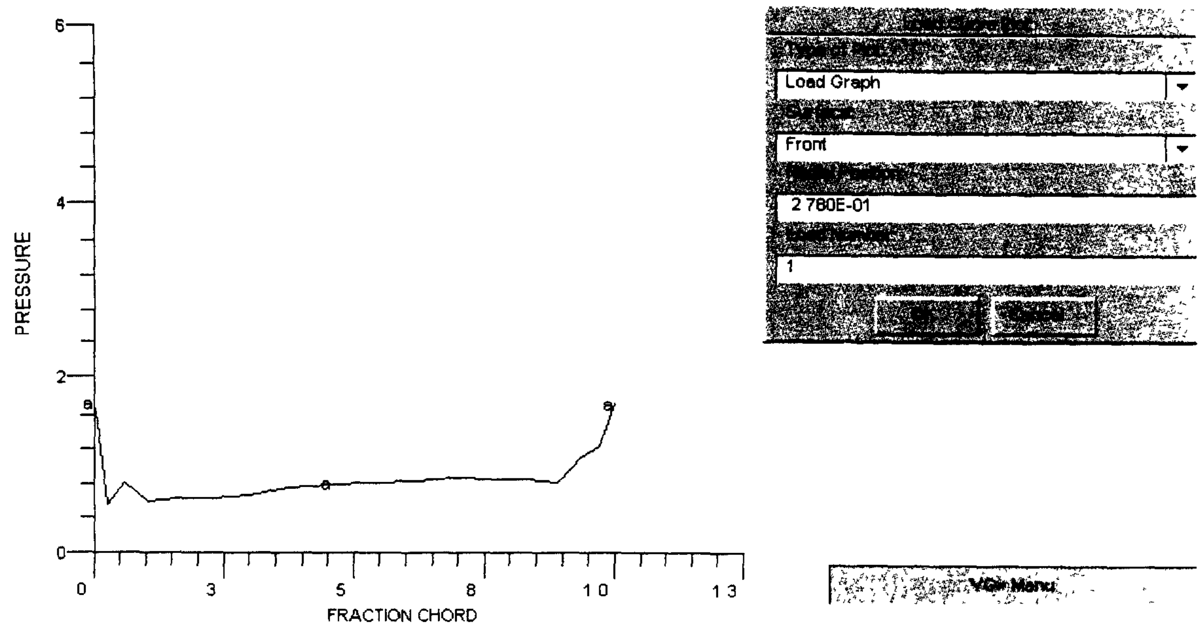


Figure 5.2: Load Graph of Pressure Distribution

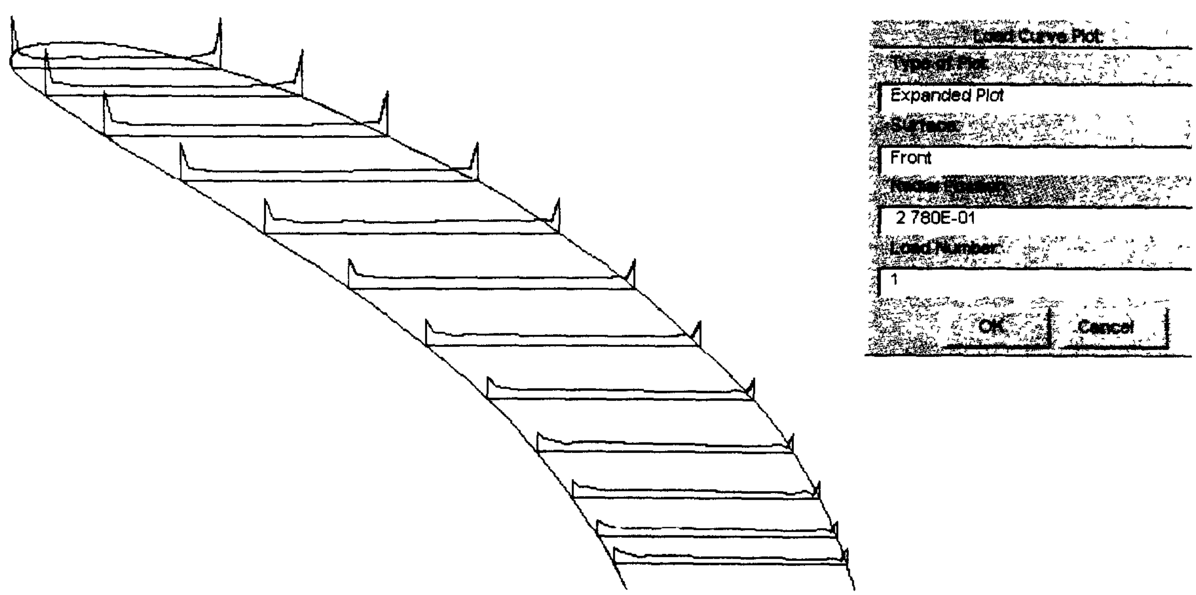


Figure 5.3: Expanded Plot of Pressure Distribution

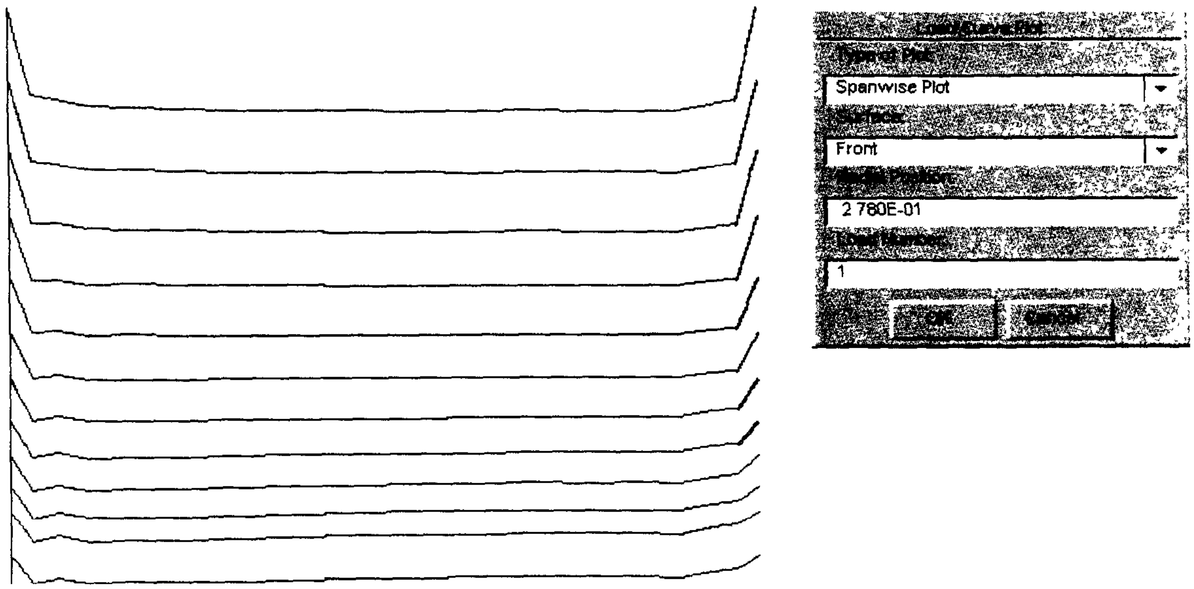


Figure 5.4: Spanwise Plot of Pressure Distribution

5.2.4 Nodal Pressures and Drag Forces

This step is performed to transform the propeller blade pressure data into a form that is acceptable by the module for generating the **Loads on FE Model.** If the adjusted pressure (PREFIX.LAD) file is available, it will be used for the computations, otherwise the data on the PREFIX.LIN file will be used. The procedure is as follows:

Click **Create/Modify** on the main menu.
Click **Nodal Pressures and Drag Forces**

At this point, a pick menu appears for selecting the print option. Select **Yes** or **No** and click **OK** when finished.

A message indicating the transformation has been completed will be displayed when finished.

5.2.5 Finite Element Mesh

This step is performed to generate the finite element mesh. The procedure is as follows:

Click **Create/Modify** on the main menu.
Click **Blade FE Model** on the **Create/Modify** submenu. Figure 5.5 shows the data entered and Figure 5.6 shows a fill plot of the finite element mesh using the general plotting capability.

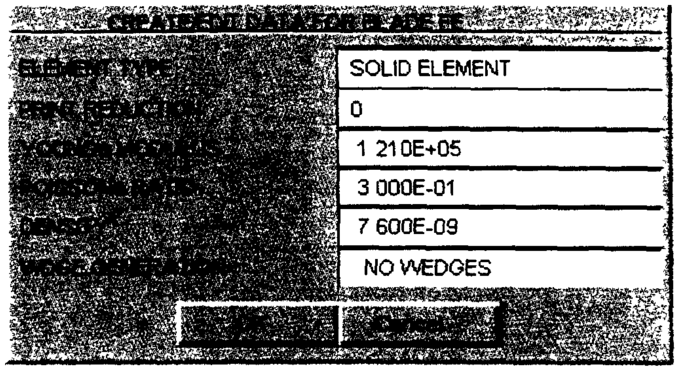


Figure 5.5: FE Model Data for P5363 Propeller Stress Analysis

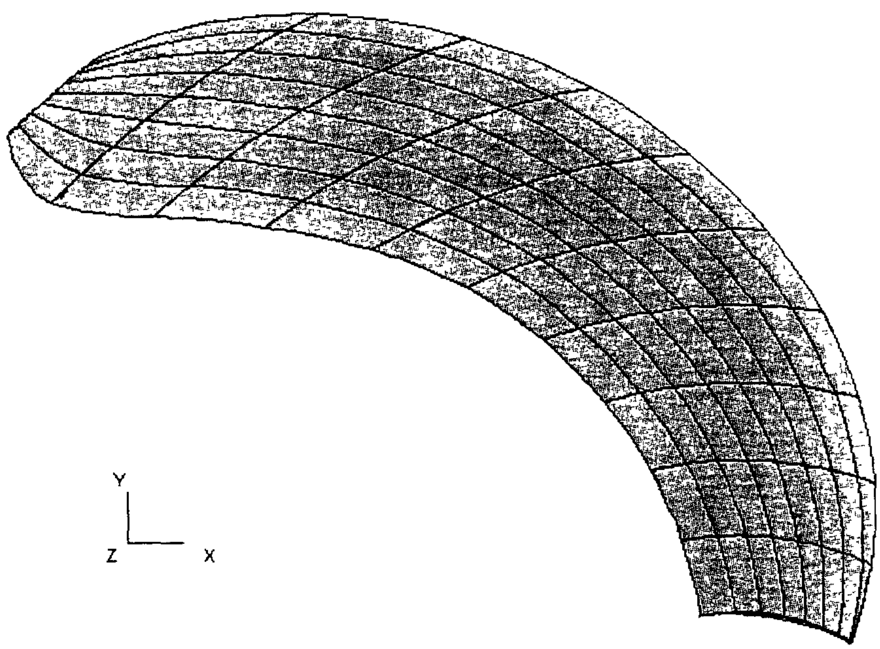


Figure 5.6: FE Mesh of P5363 Propeller Stress Analysis

5.2.6 Loads on Finite Element Model

This step involves the generation of loads on the finite element mesh. No additional data is required. The following procedure is followed:

Click **Create/Modify** on the main menu and then
Click **Loads on FE Model** on the **Create/Modify** submenu.

The program will generate the loads and provide a message when finished.

5.2.7 Boundary Conditions

Click **Create/Modify** on the main menu and then
Click **Boundary Conditions** on the **Create/Modify** submenu.

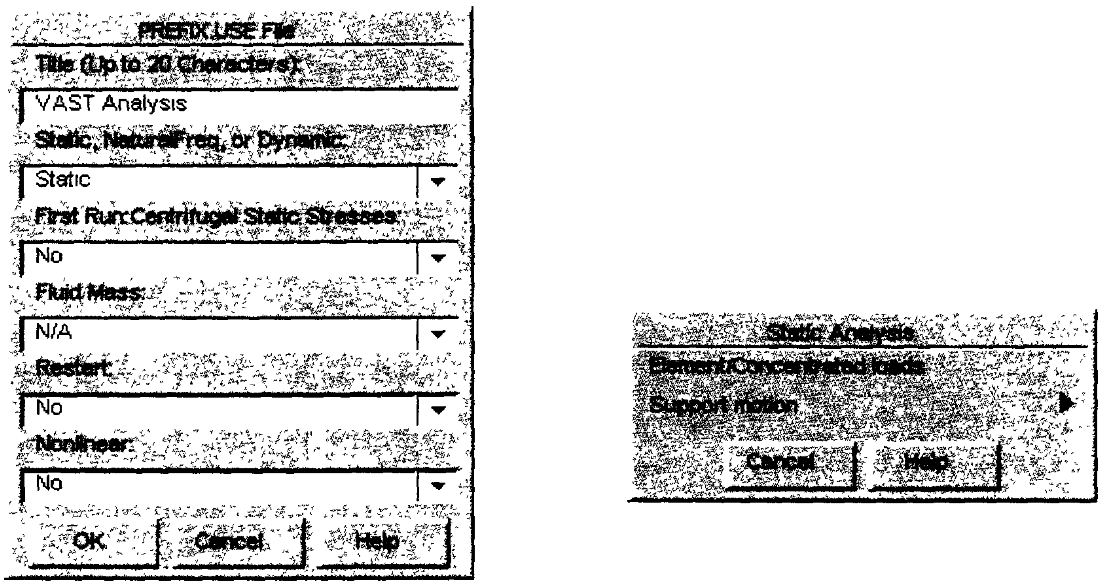
The default boundary conditions, as shown in Figure 3.22, are used for the analysis.

5.2.8 VAST Input File Generation

This step involves generating the input file for a VAST finite element analysis.

Click **Create/Modify** on the main menu and select the **VAST Input File** on the **Create/Modify** submenu. (Figure 5.7).

Select **Static** analysis, as shown in Figure 5.7. Click **OK**.



(a) (b)

Figure 5.7: Generating VAST Input File for Stress Analysis

5.3 Static Stress Analysis

The steps for performing the VAST stress analysis are the same as those described in Section 3.3.

Click **Analysis** on the main menu. Then click **Start VAST Analysis** on the **Analysis** submenu as shown in Figure 5.8.

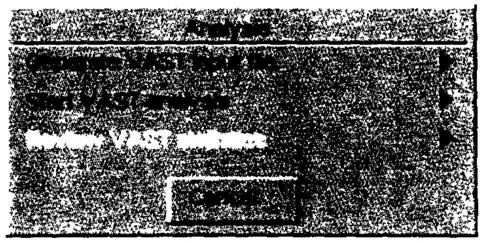


Figure 5.8: Analysis Submenu

The VAST analysis status, shown in Figure 5.9, will now be displayed. Click **Start** to start the VAST stress analysis. The progress of the analysis will be shown in the dialog box, and a message will be provided to signify the end of the analysis. Click **OK** to return to the PVAST main menu.

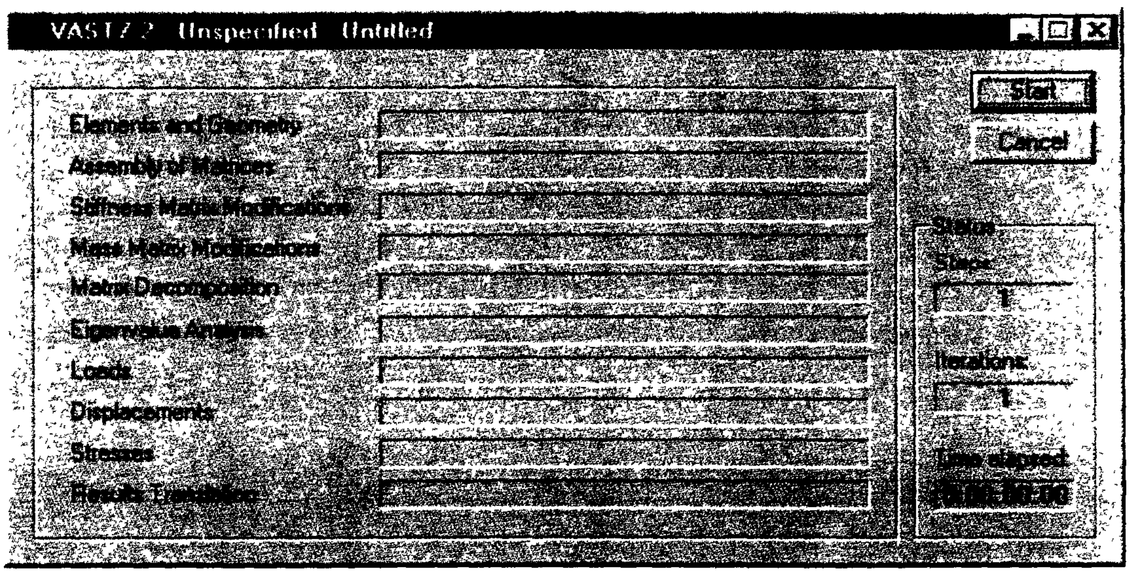
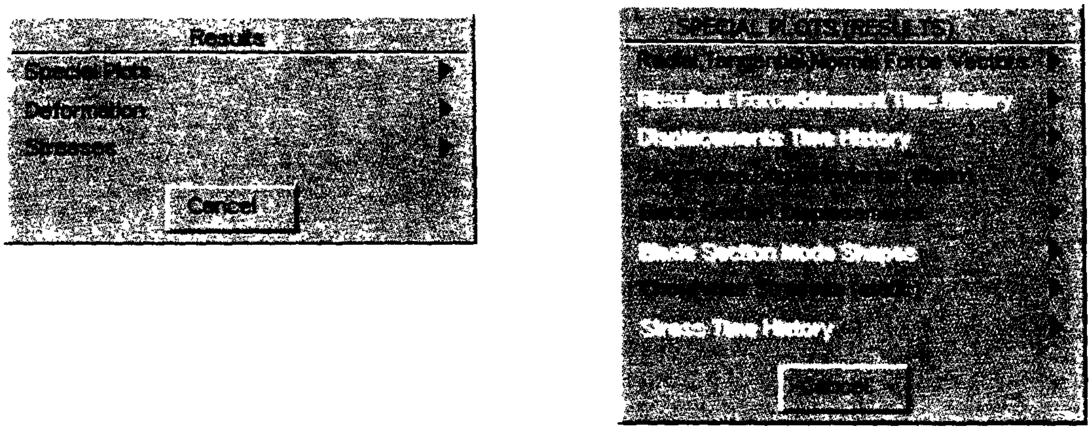


Figure 5.9: VAST Analysis Status Dialog Box

5.4 Post-Processing of Results

5.4.1 Special Plots

Click **Results** on the main menu (Figure 5.10 (a) appears).
Click **Special Plots** on the **Results** submenu (Figure 5.10 (b) appears).



(a) (b)

Figure 5.10: Menus for Selecting Special Plots Stress (Results)

To plot the radial, tangential and normal force vectors, click **Radial, Tangential, Normal Force Vectors**. Figure 5.11 shows the radial force vectors at the front.

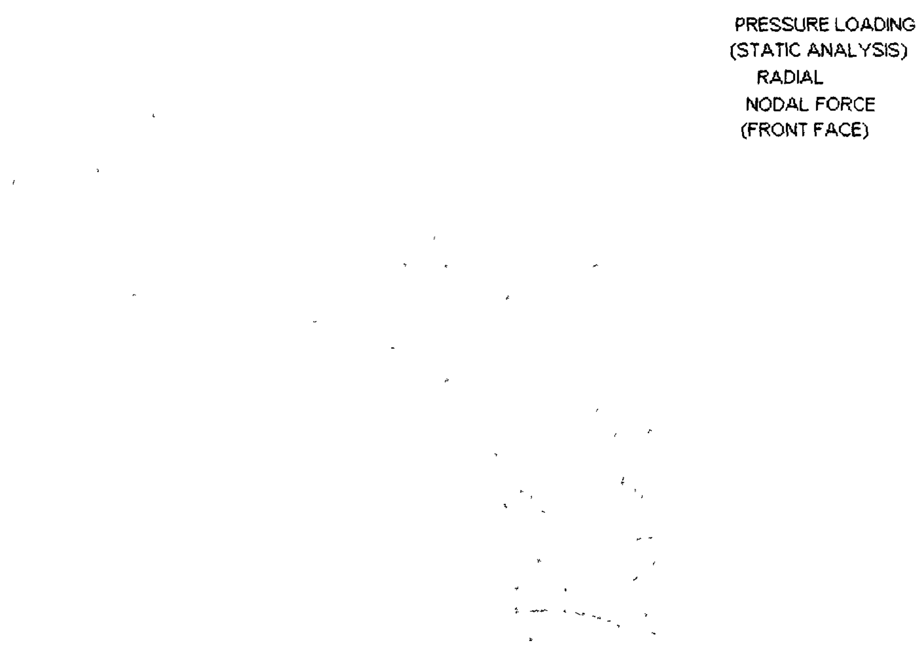


Figure 5.11: Radial Force Vectors at Front Face of P5363 Propeller Blade

To plot other components, click **Options** at the top right hand corner and toggle through the options under **Component** and **Face** to select the component and the face, as shown in Figure 5.12. Following this procedure, other components have been plotted. Figures 5.13 and 5.14 show the front face tangential and normal force vectors, respectively.

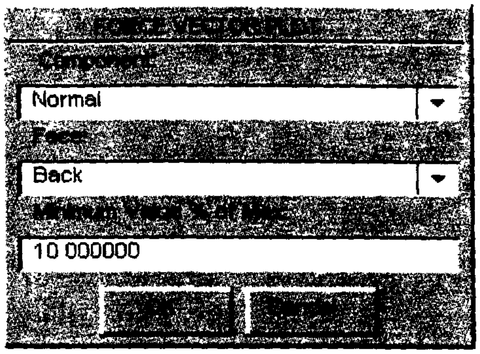


Figure 5.12: Selecting Force Components for P5363 Propeller

PRESSURE LOADING
(STATIC ANALYSIS)
TANGENTIAL
NODAL FORCE
(FRONT FACE)

Figure 5.13: Tangential Force Vectors at Front Face of P5363 Propeller Blade

Figure 5.14: Normal Force Vectors at Front Face of P5363 Propeller Blade

To plot the chordwise displacements, click **Chordwise Displacements (Static)**. A list of the radial fractions is then presented. Select the radial function at which displacement plotting is desired by clicking on the radial fractions, and click **OK**. Figure 5.15 shows plot at radial fraction 0.9155 displacement components and radial. Other positions can be plotted by clicking **Options** at the top right hand corner and using the data boxes shown in Figure 5.16.

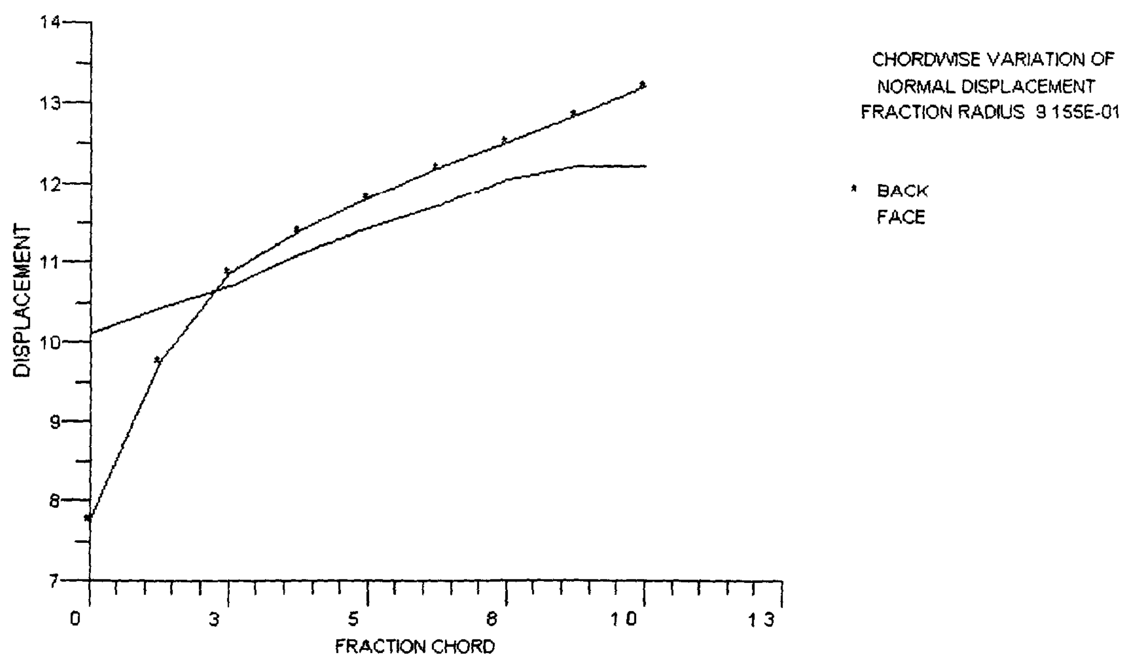


Figure 5.15: Chordwise variation of Normal Displacement of P5363 Propeller

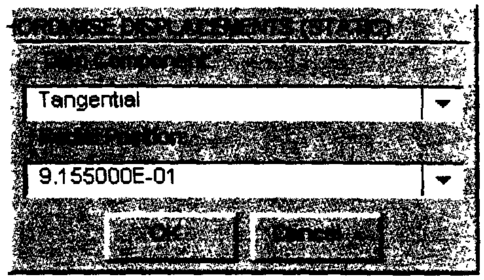


Figure 5.16: Selecting Components and Radial Positions for Plotting Chordwise Displacements in P5363 Blade

To plot the blade section displacements, click **Blade Section Displacements**. A list of the radial positions will now be displayed. Click on the appropriate radial position to select the radial position. Figure 5.17 shows the blade section displacement at radial position 0.836. Other positions can be selected by clicking options on the top right hand corner.

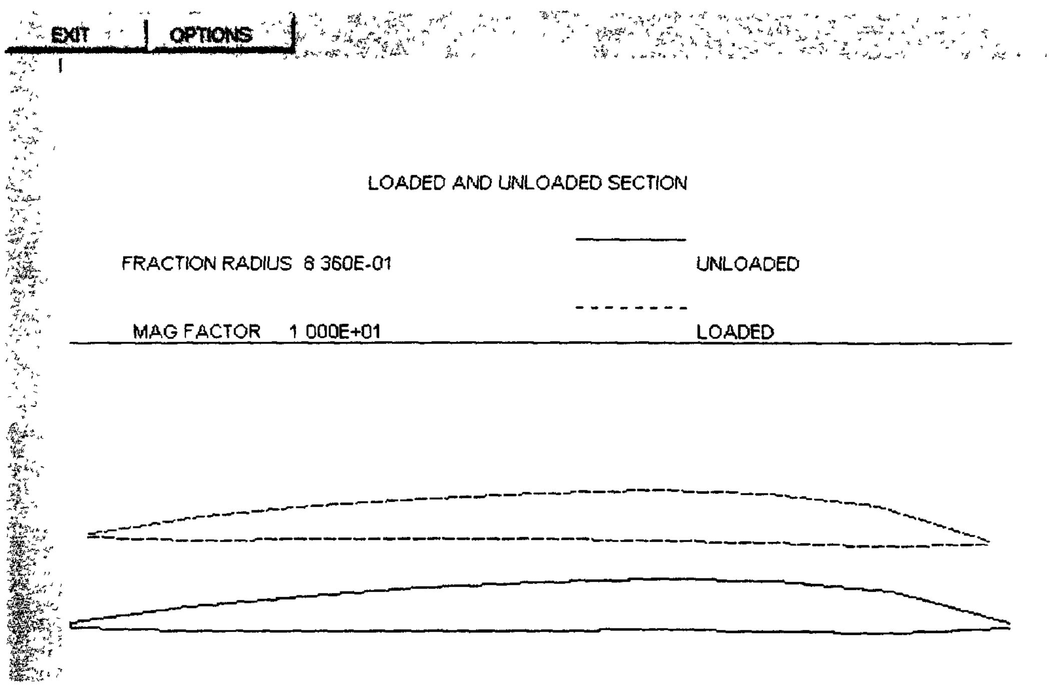


Figure 5.17: : Blade Section Displacement at 0.836 Radial Fraction in P5363 Propeller Blade

To plot the chordwise variation of stresses, click **Chordwise Stresses (Static)** on the **Special Plots (Results)** submenu. Again, the radial positions are displayed. Click the desired radial position. Figure 5.18 shows the chordwise variation of the primary principal stresses at radial function of 0.518. Other stress components and radial positions can be selected as described above.

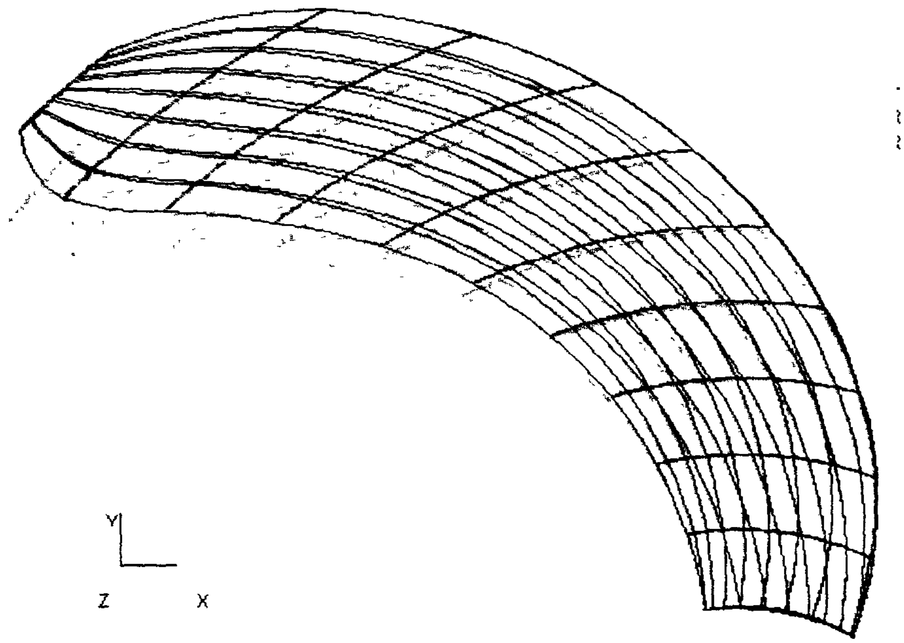


Figure 5.21: Deformed Shape Superposed with the Undeformed Shape

Plotting Contours of Mode Shapes

Plotting mode shape contours involves the following steps:

Click **Options** on the main menu
Click **Plotting** on the **Options** submenu
Click **Contour** on the **Plotting** submenu
Click **Deformation** on the **Contour** submenu
Select **Fringe** plotting

Figure 3.22 shows the contours of deformed shape.

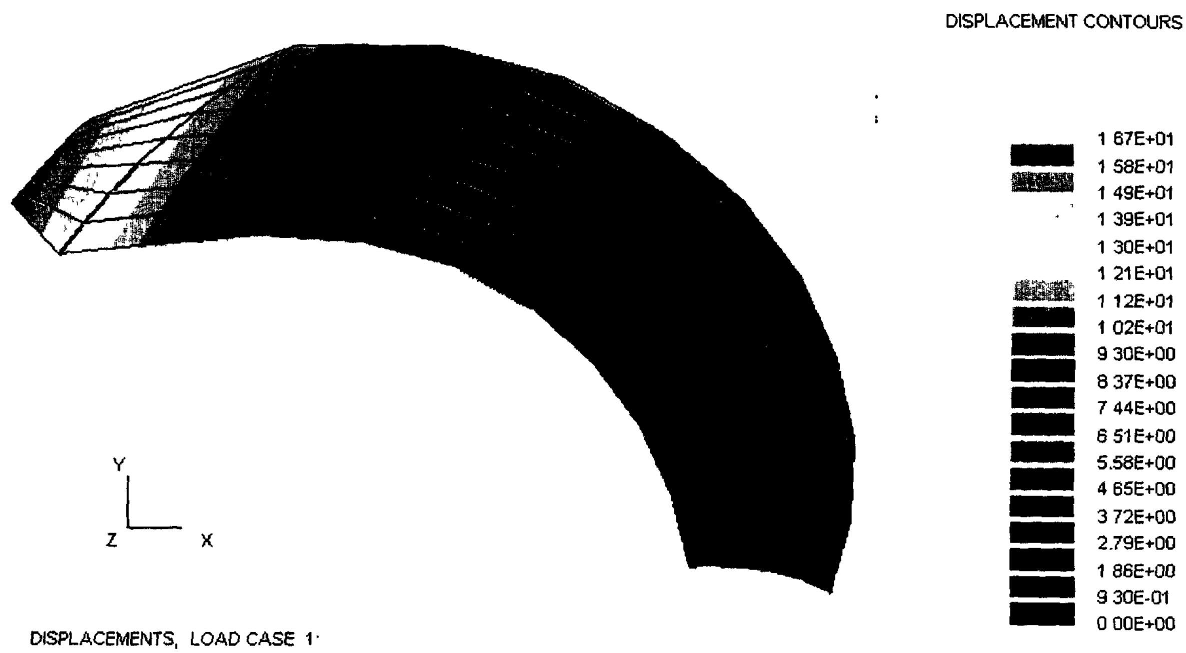


Figure 5.22: Deformation Contours of P5363 Propeller Blade

Plotting Stresses:

Click **Results** on the main menu.

Click **Stresses** on the **Results** submenu (Figure 5.19 (a)).

Accept **Transfer to Database** (for first time).

Click **Static**.

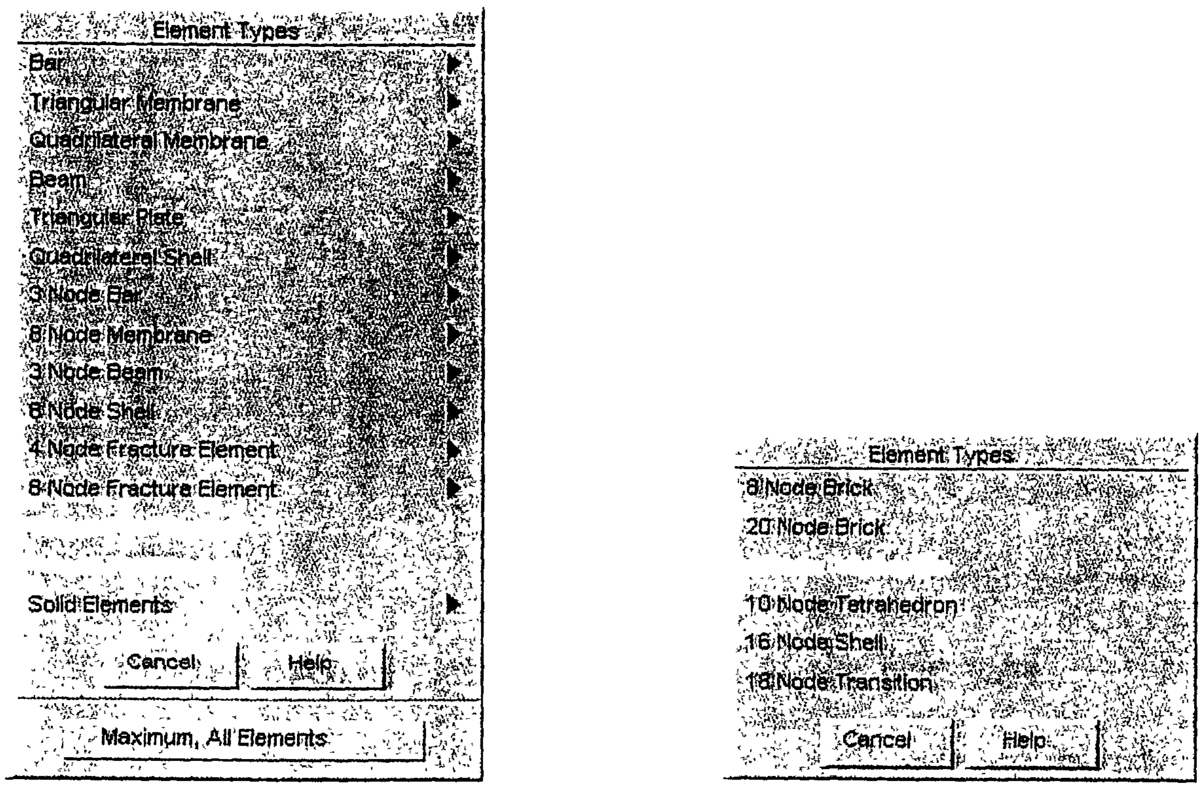
Enter load case (equal to 1 in this case)

Click **Solid Elements** on the **Element Types** submenu (Figure 5.23 (a)) and select 20-node brick (Figure 5.23 (b).

Select von Mises on **the 20-node Solid** element stress component submenu (Figure 5.24).

Click **Plot Stresses**, to plot the element stresses, as shown in Figure 5.25.

Following the procedure outlined above for plotting displacement contours, the von Mises stress contours can be plotted as shown in Figure 5.2 (b).



(a) (b)

Figure 5.23: Element Types Submenu

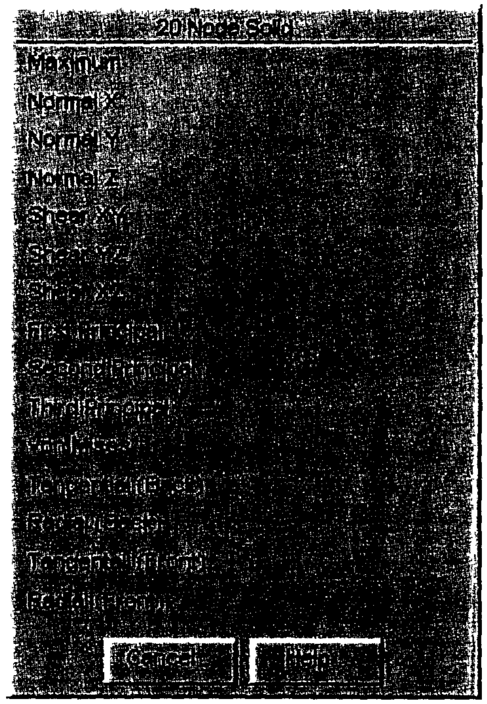


Figure 5.24: Stress Components for 20-node Solid

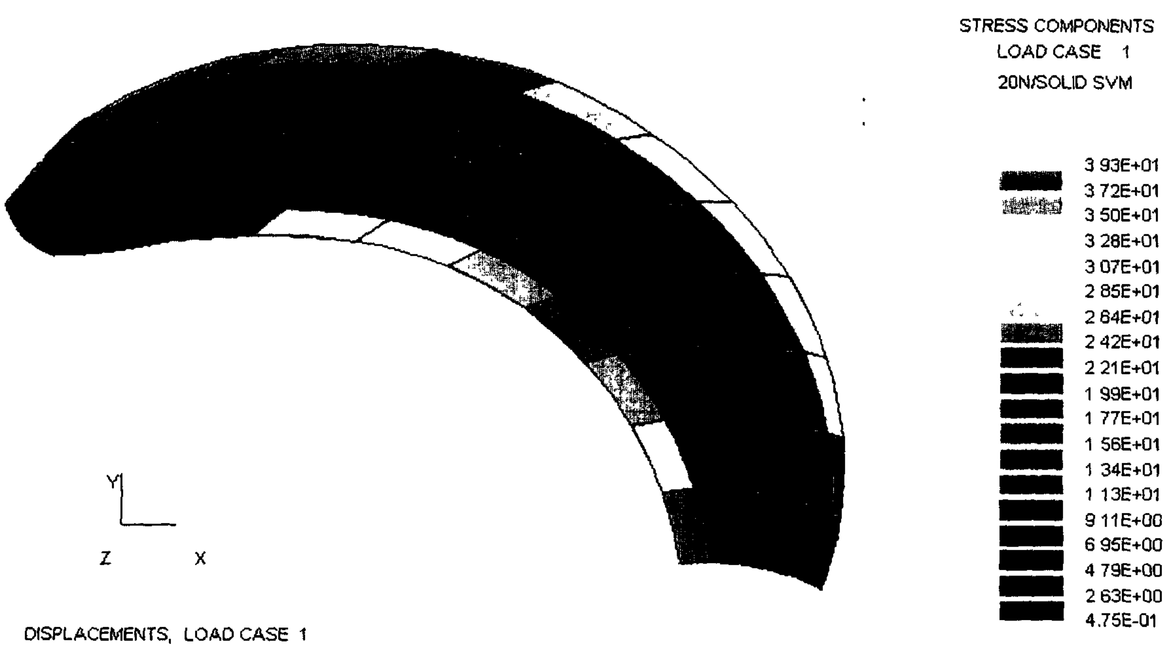


Figure 5.25: von Mises Stresses in P5363 Propeller Blade

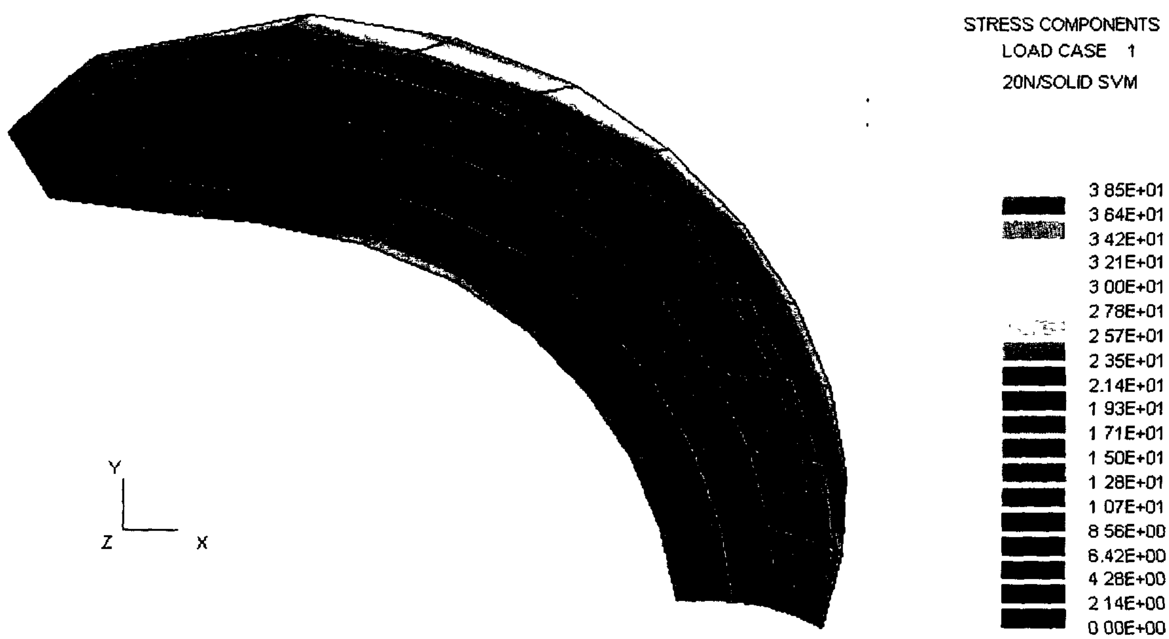


Figure 5.26: von Mises Stress Contours in P5363 Propeller Blade

6. BLADE-FILLET-HUB MODEL

6.1 Introduction

In this chapter we shall demonstrate how to generate blade-fillet-hub models. It is assumed that the blade geometry has been defined in Chapter 2. Thus a PREFIX.DAT file is available in the working directory, where PREFIX is the alphanumeric prefix of the file name. The P5363, P4388 and NRC45 propeller blades shall be used for this demonstration. Finite element grid proportions of these blades have been generated in Chapter 3. With the grid proportions already defined, the remaining steps are to:

(a) Generate the blade model;
(ii) Generate the blade finite element mesh; and
(iii) Generate the sub-structural models of the blade, fillet and hub.

It should be noted that the model type bust be set to **Blade Fillet Hub** or **Multi Blade Hub**. The **Multi-Blade Hub** model is selected for this demonstration.

6.2 Blade-Fillet-Hub Model of P5363 Propeller Blade

6.2.1 Root Model Generation

Click **Create/Modify** on the main menu.
Click **Root Model** on Create/Modify submenu.

Figure 6.1 appears for defining the root data.

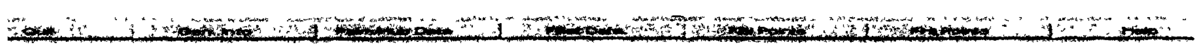
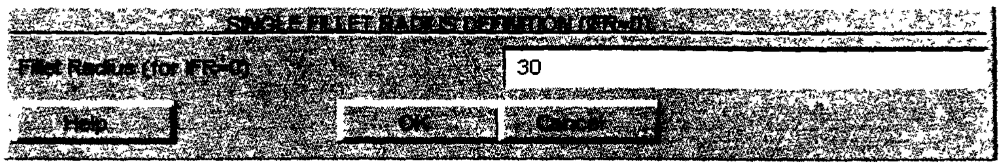


Figure 6.1: Root Model Data Generation Submenu

Figure 6.2 shows the data entered under **Gen Info.**
Figure 6.3(a) shows the data entered under **Palm/Hub Data.** Selection of the option **Create** coordinate points results in the display shown in Figure 6.3(b).
Figure 6.4 shows the data entered under **Fillet Data**

For the type of model selected, no data under FBI (fillet-blade-interface) points and FHI (fillet-hub-interface) points are required.

Pvast7 3examplesman doc



(b)

Figure 6.5: Correcting an Error in the P5363 Root Data (a) Error Message; (b) Revised Fillet Radius

It should be noted that the PVAST, root model generation algorithms assume that the data supplied is reasonable, and provide guidance on how to resolve some cases of incorrect or unreasonable data. As PVAST is designed as an analysis tool, it might not be possible to detect all kinds of inappropriate data during design iterations which might lead to the generation of badly shaped elements in the fillet and hub finite element models, or even program crashes. To avoid these situations, reasonable root data obtained by measurements from actual blades may be required.

Plotting Blade-to-Root Intersection

Click **Display** on the main menu.
Click **Special Plots (Model)** on the **Model Display** submenu.
Click **Blade-to-Root Intersection** on the **Special Plots Model** option.

Figure 6.6 shows the blade-to-root intersection. This figure aids in determining if the blade is well positioned on the hub. In the present example, the blade is seen to be positioned properly on the hub.

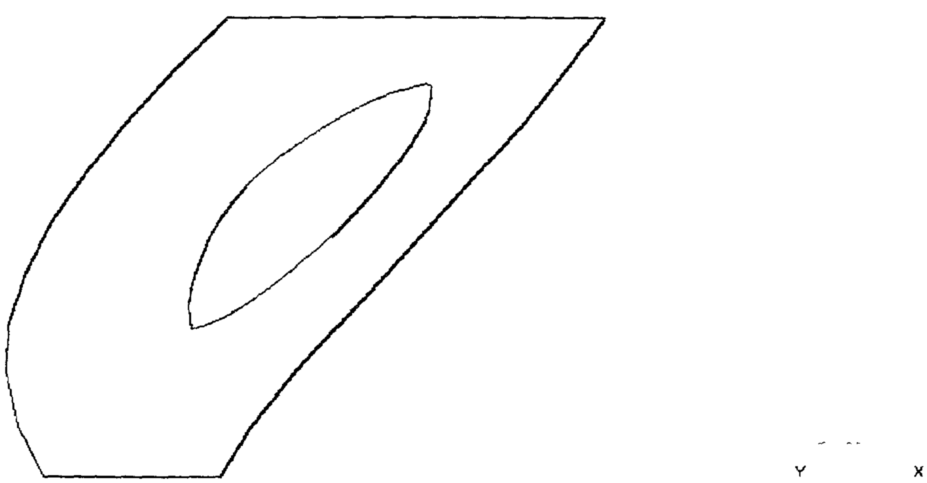
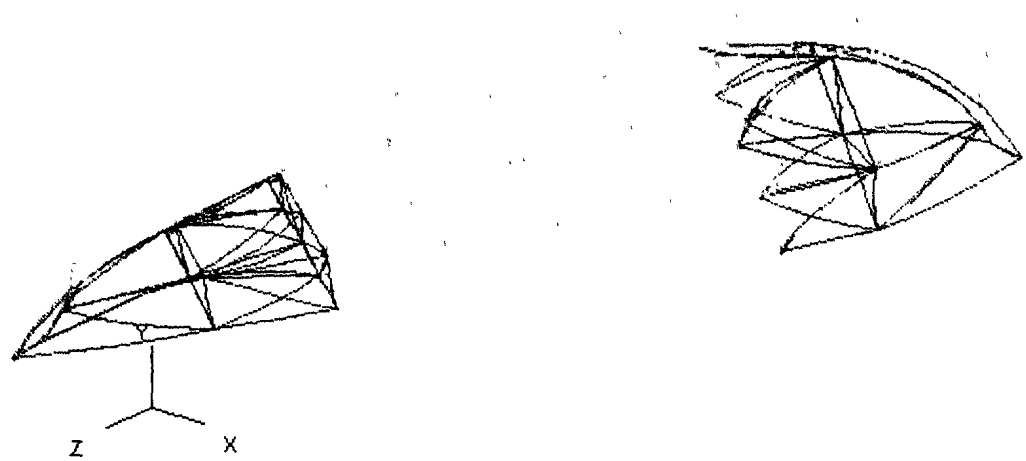


Figure 6.6: P5363 Blade-to-Root Intersection Plot

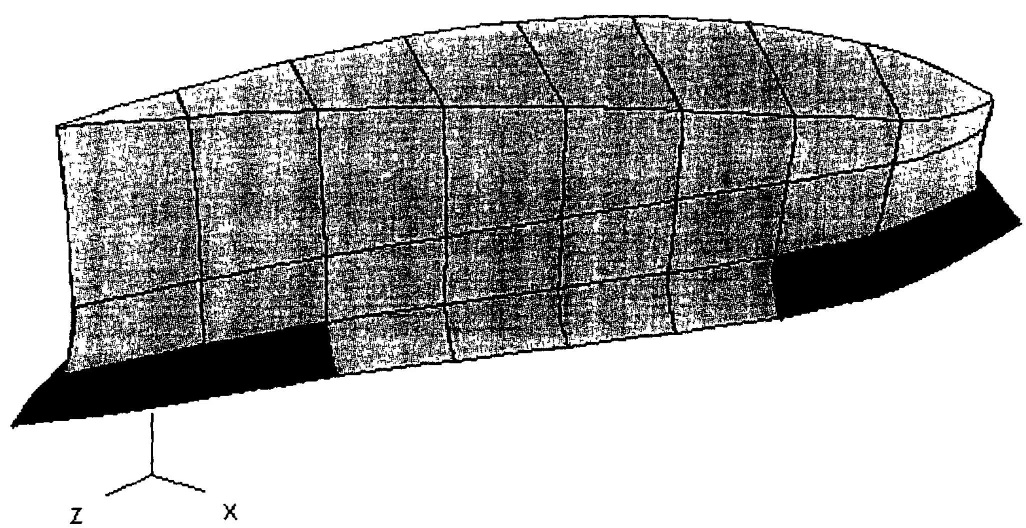
Plotting Fillet and Hub Models

Click **Display** on the main menu.
Click **Root FE Model** on the **Model Display** submenu.
Click **Fillet** on the **Root Model** submenu to plot the fillet or hub model.

Figures 6.7 (a) and (b) show the wire frame and fill plot models of the fillet finite element model.



(a)



(b)

Figure 6.7: P5363 Fillet Finite Element Models, (a) Wire Frame, (b) Fill Plot

Figure 6.8 shows a fill plot of the hub model.

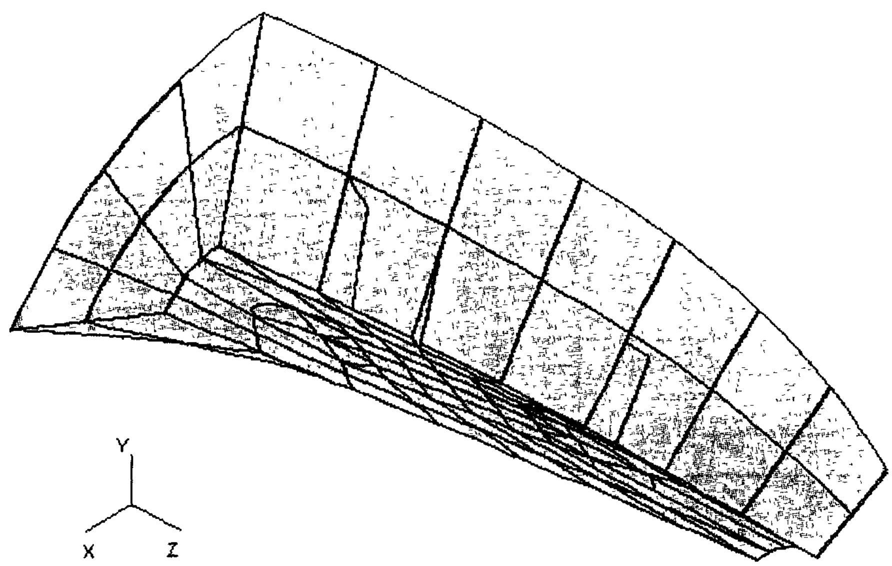


Figure 6.8: P5363 Hub Finite Element Model

6.2.2 Blade Mode Generation

This step is required to generate the finite element model of the blade. The procedure is similar to that described in Chapters 3 and 5. The data entered for the blade finite element generation is shown in Figure 6.9. The blade's finite element mesh is shown in Figure 6.10, appended to the fillet and hub model models.

CREATE/EDIT DATA FOR BLADE FE

ELEMENT TYPE	SOLID ELEMENT
PRINT REDUCTION	0
YOUNGs MODULUS	1 210E+05
POISSONs RATIO	3 000E-01
DENSITY	7 600E-09
WDGE GENERATION	NO WEDGES

OK Cancel

Figure 6.9: Blade FE Data for P5363 Blade-Fillet-Hub Model

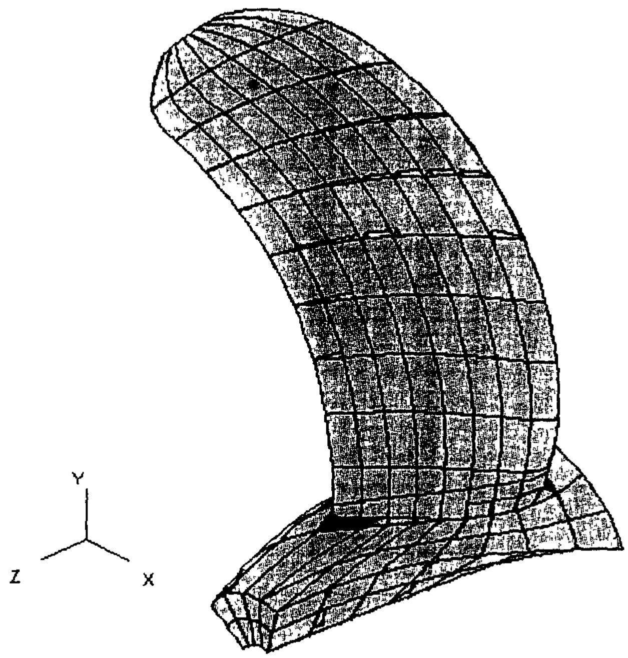


Figure 6.10: P5363 Blade FE Model Appended to Fillet and Hub Models

6.2.3 Substructured Blade-Root-FE Model

Click **Create/Modify** on the main menu.

Click **Substructured Blade-Root FE** on the **Create/Modify** submenu. The substructured data entry submenu now appears as shown in Figure 6.11.

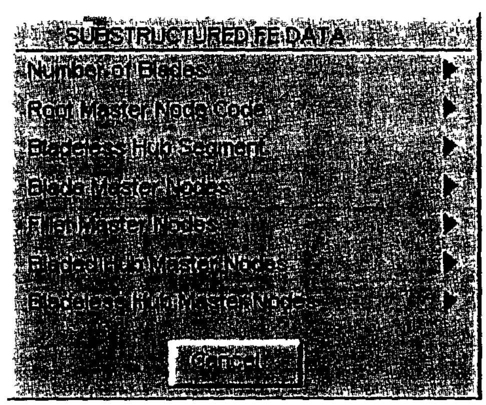


Figure 6.11: Defining Substructured FE Data

Click **Number to Blades** to define the number of blades. For this demonstration, the number of blades is 5.

Click **Root Master node Code** to define how the root master nodes are defined. The **Interface and Side Nodes** option is selected.

Click **Bladeless Hub Segment** to specify if a bladeless hub segment is to be used. Select **No**.

Click Blade Master nodes to define master nodes for each of the five blades. Select **No Additional Nodes** for each of the five blades. This is the default option in which the blade master nodes are automatically defined by PVAST.

Click **Fillet Master Nodes** to define master nodes for the five fillets. Again the default option (No Additional Nodes) is selected.

Click **Bladed Hub Master Nodes** to define master nodes for the five hubs. Again the default nodes are used (**No Additional Nodes**).

Since no bladeless hub was selected, it is not necessary to define data for the **Bladeless Hub Master Nodes.**

Note that the other options of master node generation (i.e. **Create Additional Nodes** and **Use Previous Additional Nodes** are not functional in this current version of PVAST.

Click Cancel to exit the substructure data generation. The program now generates the substructured model of the blade-fillet and hub. Figure 6.12 shows the five bladed substructured model.

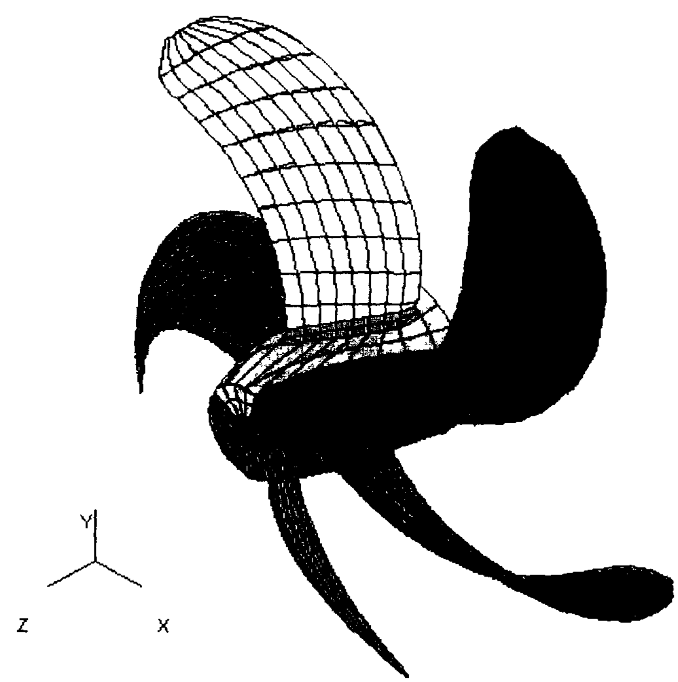


Figure 6.12: Five-Bladed Substructural Model of P5363 Propeller Blade

7. BLADE-FILLET-PALM MODEL

7.1 Introduction

This chapter illustrates how to develop a blade-fillet-palm model. Another DREA propeller blade designated as TRUMP is used for the demonstration because some reasonable palm root model data exists for the blade. The starting point is the PREFIX.DAT file containing the blade geometric data. The steps for generating the blade-fillet-palm model involve:

- Selection of blade geometry;
- Generation of blade geometry
- Definition of FE grid proportions
- Generation of the root model;
- Generation of the blade finite element model; and
- Generation of substructured model.

However, emphasis is placed on the last three steps, as other steps have been described in detail in previous sections.

7.2 Generation of the Blade-Fillet-Palm Model

7.2.1 Selection of Model Type

Click model on the main menu
Select **Blade Fillet Palm (Bolts as Rods)** or **Blade Fillet Palm (Superelements)** on the main **PVAST Model** submenu.

In this demonstration the Blade **Fillet Palm (Superelements)** option is selected as shown in Figure 7.1. For this type of model, the bolt holes are treated as bolt hole superelements. For the Blade Fillet Palm (Bolt as Rods) model, the bolts are treated as rods. *It should, however, be noted that the options to generate the bolts and bolt hole superelements have not been available in this version of PVAST.*

Pvast7 3examplesman doc

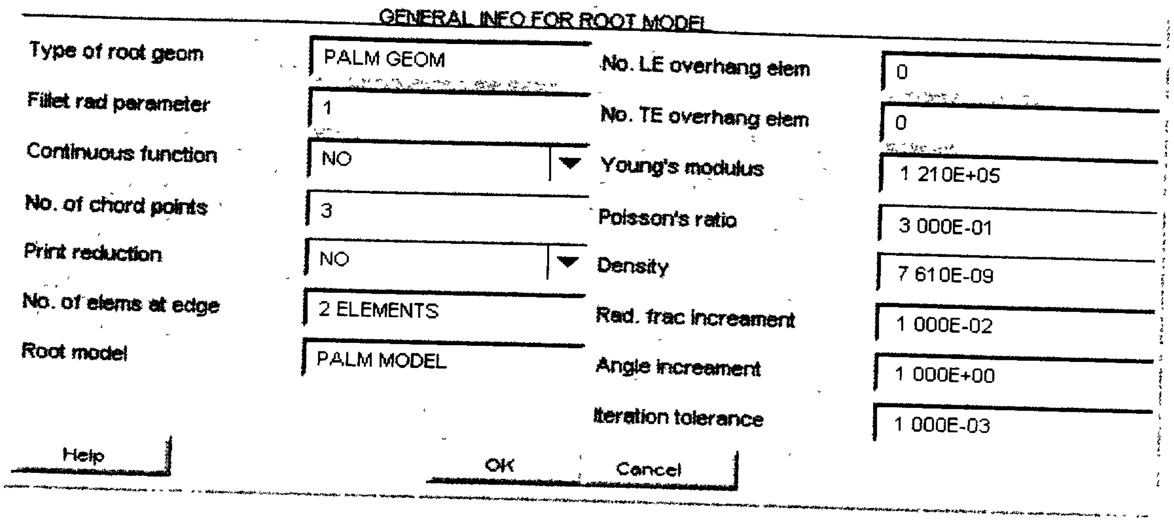


Figure 7.4 (a):

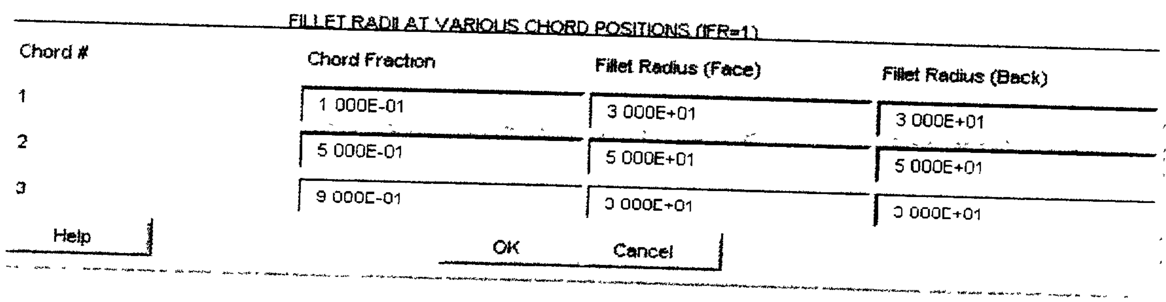


Figure 7.4 (b)

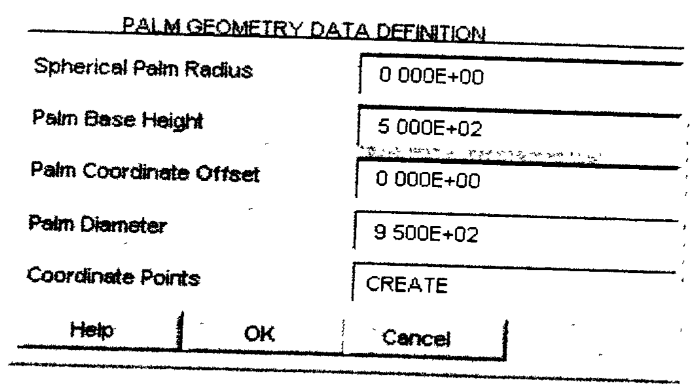


Figure 7.4 (c):

X, Y, Z COORDS FOR PALM GEOMETRY DATA DEFINITION

Coord No	X Coord	Y Coord	Z Coord
1	3 520E+02	5 800E+02	-4 300E+02
2	1 540E+02	5 800E+02	-5 280E+02
3	-1 540E+02	5 800E+02	-5 280E+02
4	-3 520E+02	5 800E+02	-4 300E+02
5	5 280E+02	5 800E+02	-1 540E+02
6	1 760E+02	7 620E+02	-1 760E+02
7	-1 760E+02	7 620E+02	-1 760E+02
8	-5 280E+02	5 800E+02	-1 540E+02
9	5 280E+02	5 800E+02	1 540E+02
10	1 760E+02	7 620E+02	1 760E+02
11	-1 760E+02	7 620E+02	1 760E+02
12	-5 280E+02	5 800E+02	1 540E+02
13	3 520E+02	5 800E+02	4 300E+02
14	1 540E+02	5 800E+02	5 280E+02
15	-1 540E+02	5 800E+02	5 280E+02
16	-3 520E+02	5 800E+02	4 300E+02

Help OK Cancel

Figure 7.4 (d):

Figure 7.5 shows the blade-to-root intersection which is plotted through **Display/Special Plots (Model) ; Blade-to-Root Intersection.**

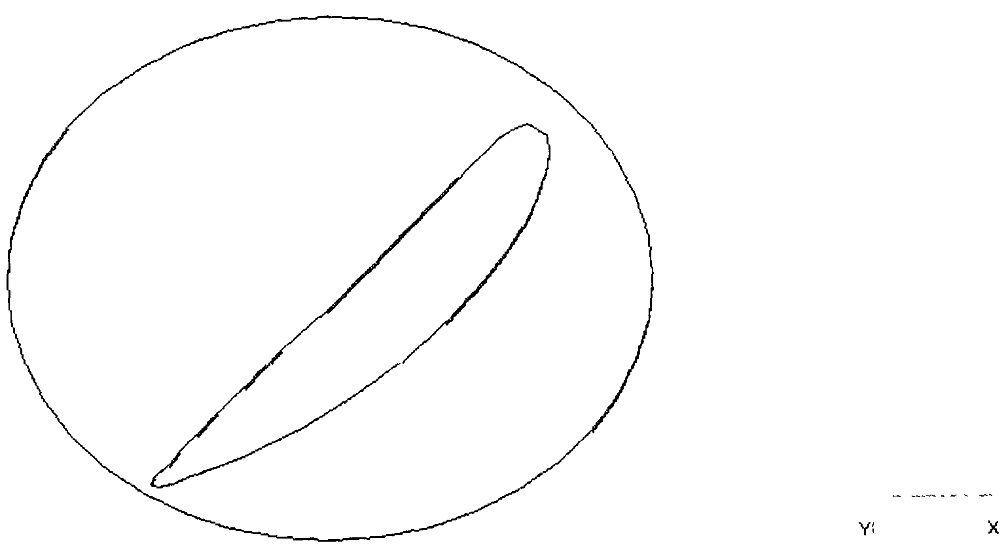


Figure 7.5: Blade-to-Root Intersection of TRUMP Propeller

Figure 7.6 shows the fillet model using the general purpose wire frame and fill plot options.

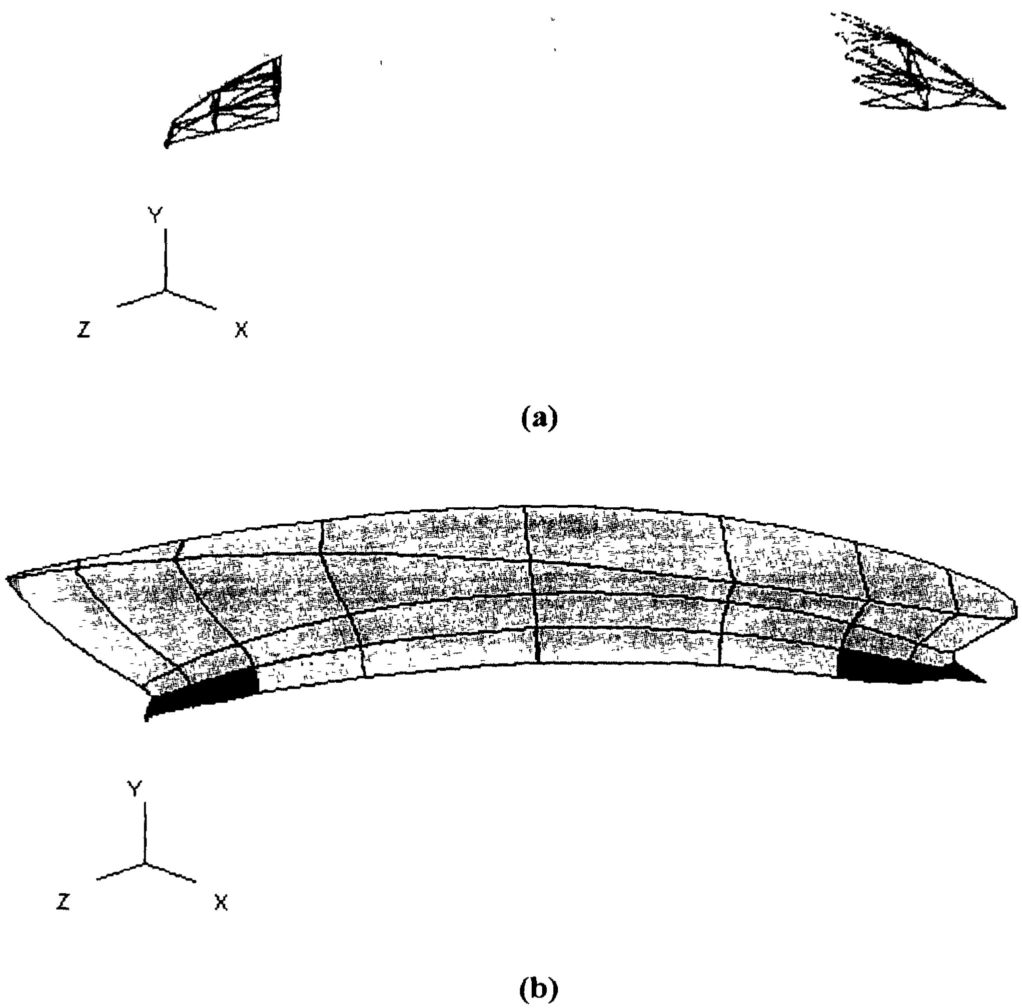


Figure 7.6: Finite Element Model of Fillet of TRUMP Propeller, (a) Wire Frame Plot and (b) Fill Plot

Figure 7.7 shows the palm model using the general purpose fill plot options.

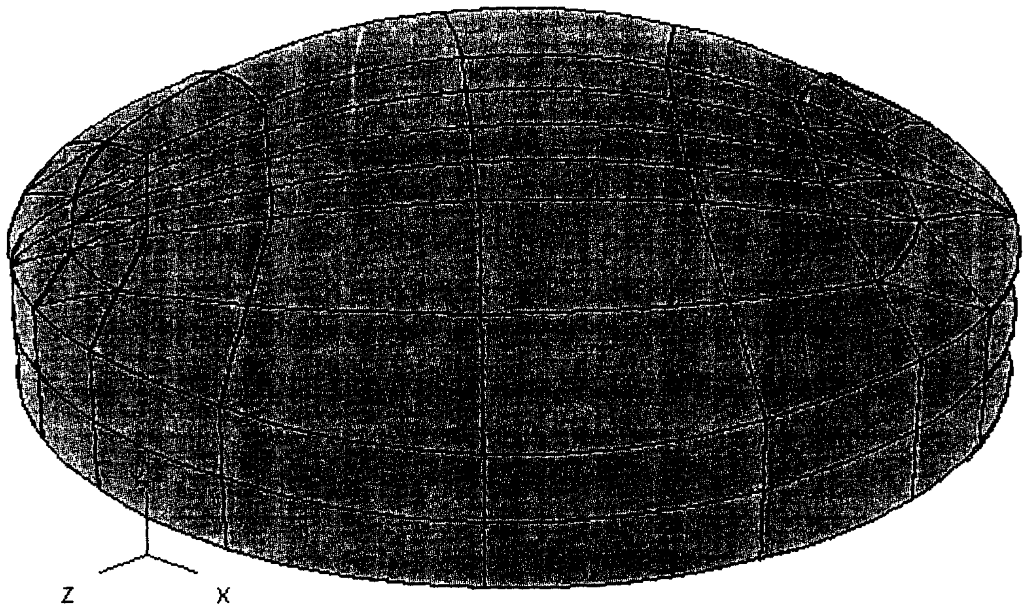


Figure 7.7: Finite Element Model of Palm of TRUMP Propeller

7.2.5 Blade Model Generation

Figure 7.8 shows the submenu for generating the blade finite element model, and Figure 7.9 shows the finite element mesh generation using the general purpose plotting option.

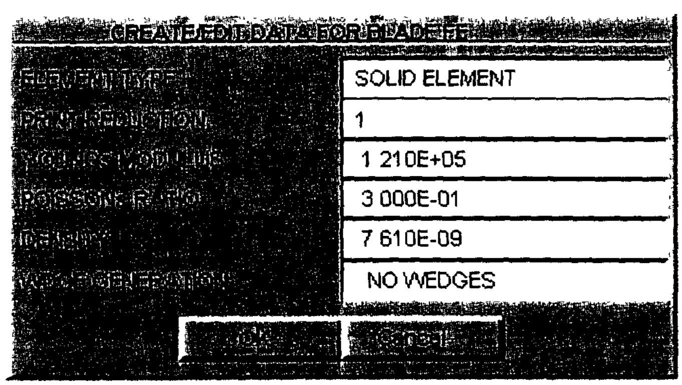


Figure 7.8: Submenu for Generating Blade Finite Element Model

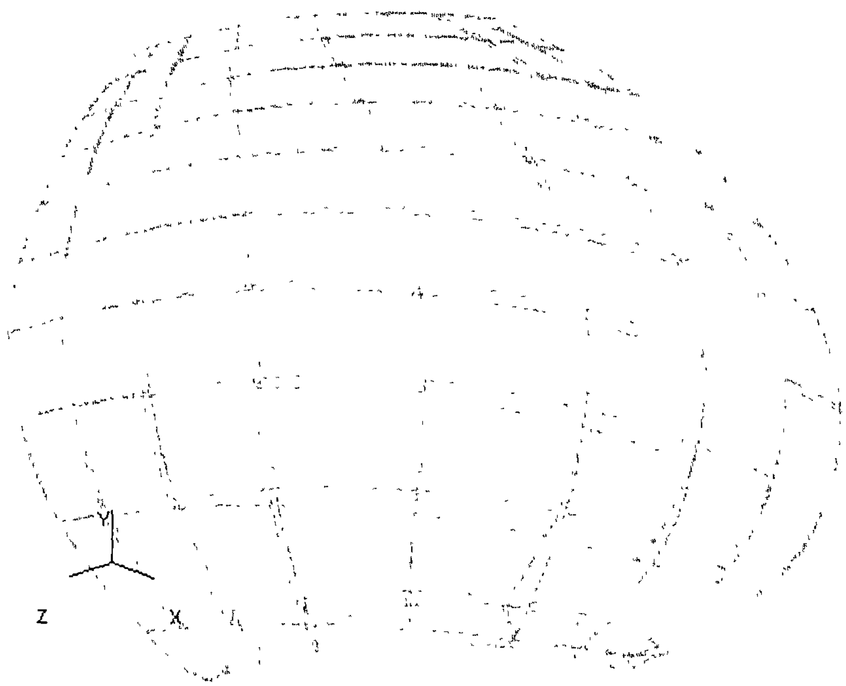


Figure 7.9: Blade FE Model of TRUMP Propeller

7.2.6 Generation of Substructured Model

To generate the substructured model click **Create/Modify** and then **Substructured Blade Root FE** and enter the required data. in the present examples, no additional data was entered and the default values have been used throughout. The model generated is displayed in Figure 7.10.

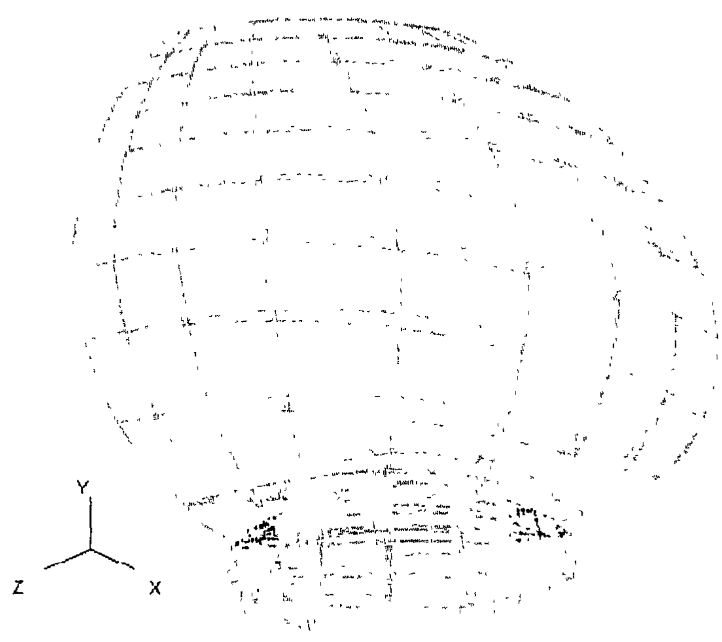


Figure 7.10

8. TWO-DIMENSIONAL HUB MODEL WITH KEYWAY

8.1 Introduction

This chapter demonstrates how to generate a keyway model. The TRUMP propeller blade is used for the demonstration. The blade geometry, finite element and root models have been presented in Chapters 2 and 7. The steps involved include:

- Generation of the keyway model,
- Performance of the analysis, and
- Post processing of results.

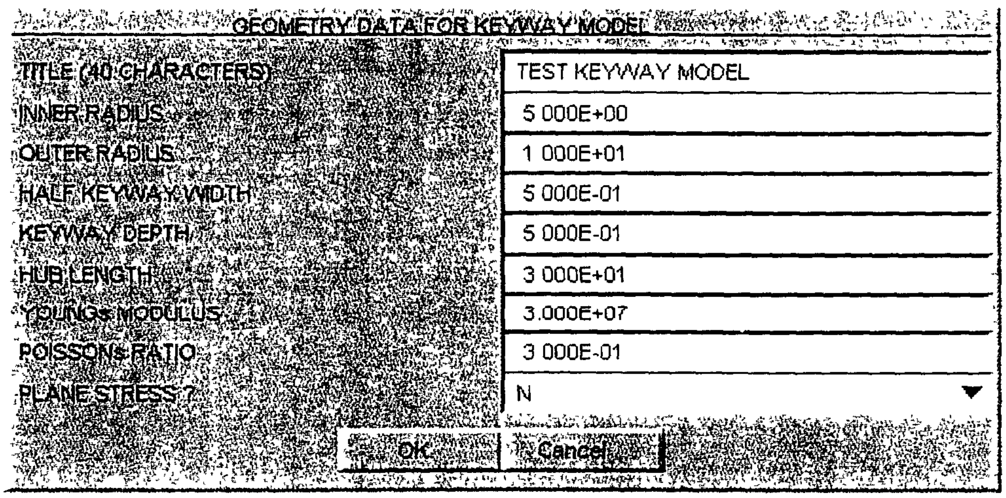


Figure 8.1: Keyway Model Data

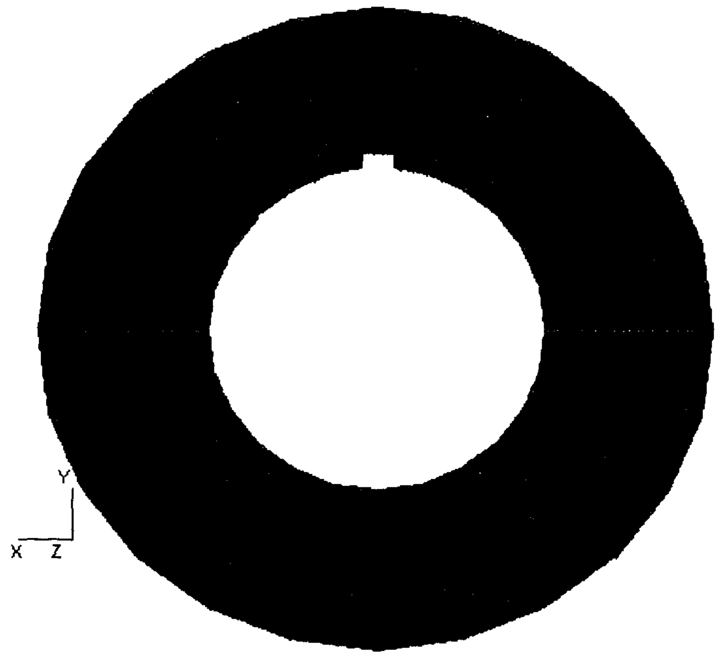


Figure 8.2: Keyway Finite Element Model

8.2 Finite Element Model Generation

Figure 8.1 shows the data entered for generation of the keyway model for this blade, and Figure 8.2 shows the keyway finite element model generated. Figure 8.3 shows the keyway load data.. Figure 8.4 shows the skew coordinates.

LOAD DATA FOR KEYWAY MODEL

NUMBER OF BLADES	5
ANGLE (DEG)	5 000E+00
TANGENTIAL FORCE	1 000E+01
OVERTURNING MOMENT	50
RADIAL FORCE	30

OK Cancel

Figure 8.3: Keyway Load Data

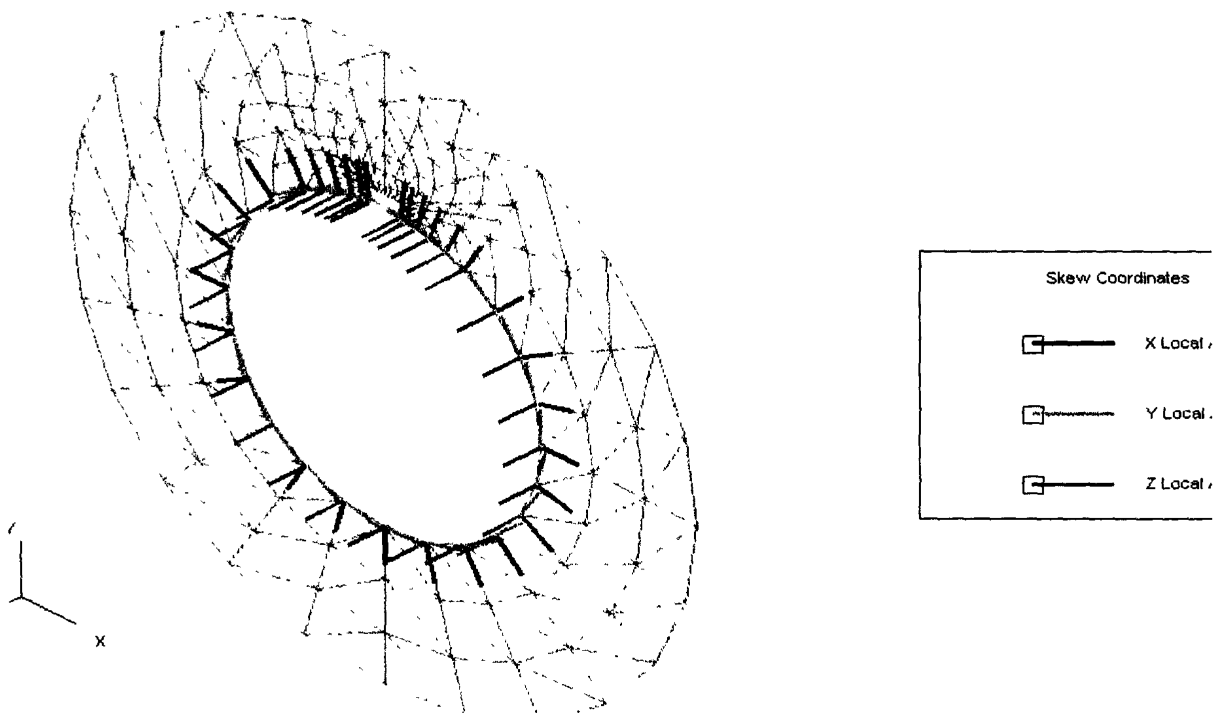


Figure 8.4: Skew Coordinates

8.3 Post-Processing Results

Figure 8.5 shows contours of the displaced shape.

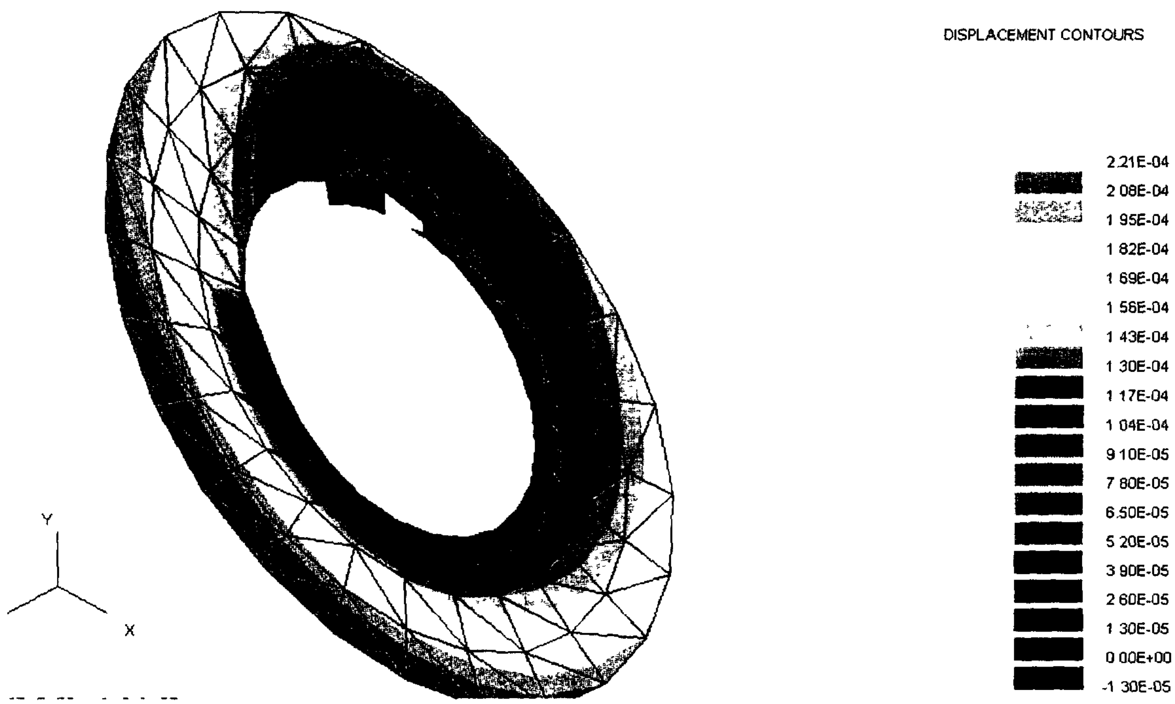


Figure 8.5: Contours of Displaced Shape

9. BLADE AIR CHANNEL STRESS ANALYSIS

9.1 Introduction

This chapter demonstrates how to generate an air channel model from a blade model and to perform a detailed stress analysis of the air channel model, using a top-down approach. The method adopted is that the blade be first modeled and analyzed, ignoring the effects of the air channel and then a detailed model of a portion of the blade containing the air channel is then developed. Thus, to generate this model, it is required that a Blade Only finite element stress analysis be performed first. The P5363 propeller blade will be used for this demonstration. A stress analysis of this blade was performed in Chapter 5, and the results will be used in the development of the air channel model. With the stress analysis of the blade already completed the remaining steps are:

- To generate the air-channel finite element model;
- To perform a stress analysis of the model; and
- To post-process the results.

These steps are described in the following sections.

9.2 Generation of Finite Element Model of Air Channel

9.2.1 Select Model Type

Click **Model** on the main menu and ensure click on the **Air Channel** model, as shown in Figure 9.1.

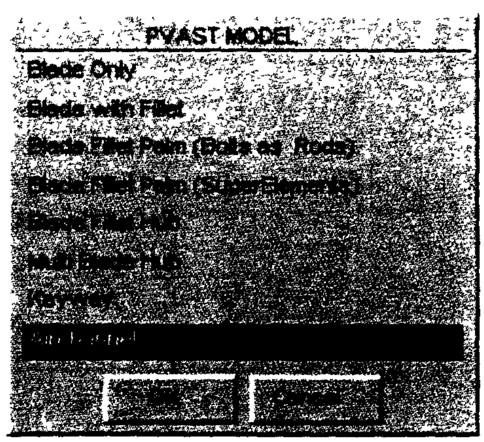


Figure 9.1: Selection of Air Channel Model

9.3 Generate Air Channel Geometric and FE Data

Click the **Create/Modify** on the main menu and select the **Generate Air Channel** option on the **CREATE/MODIFY** submenu as shown in Figure 9.2, to generate the air channel data..

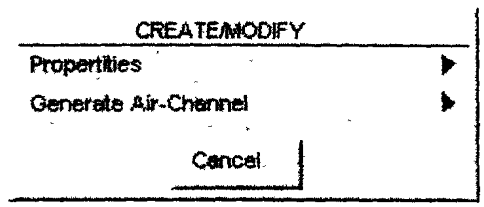
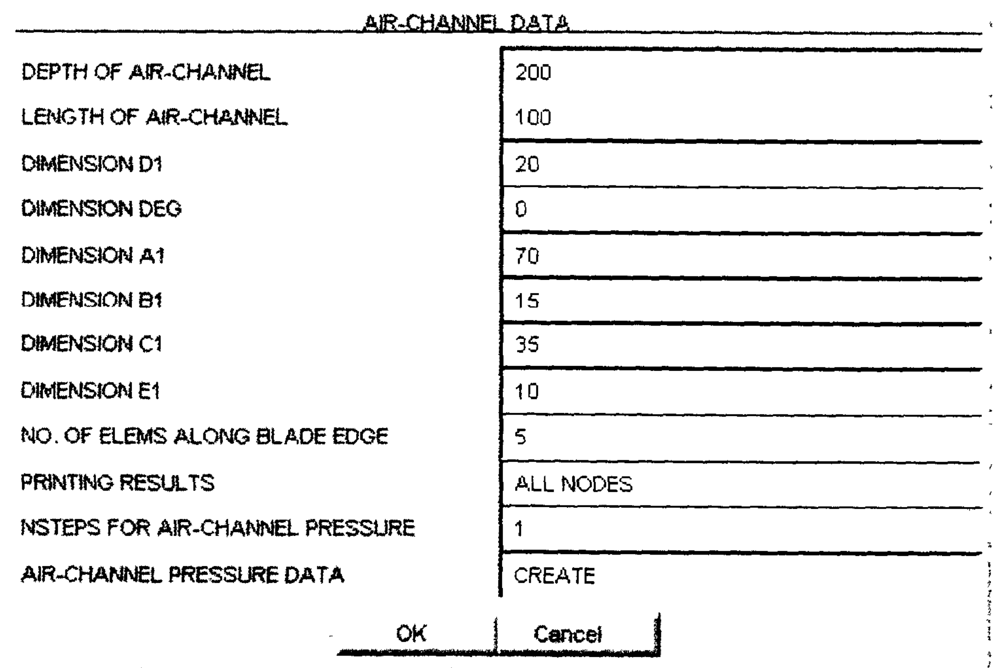
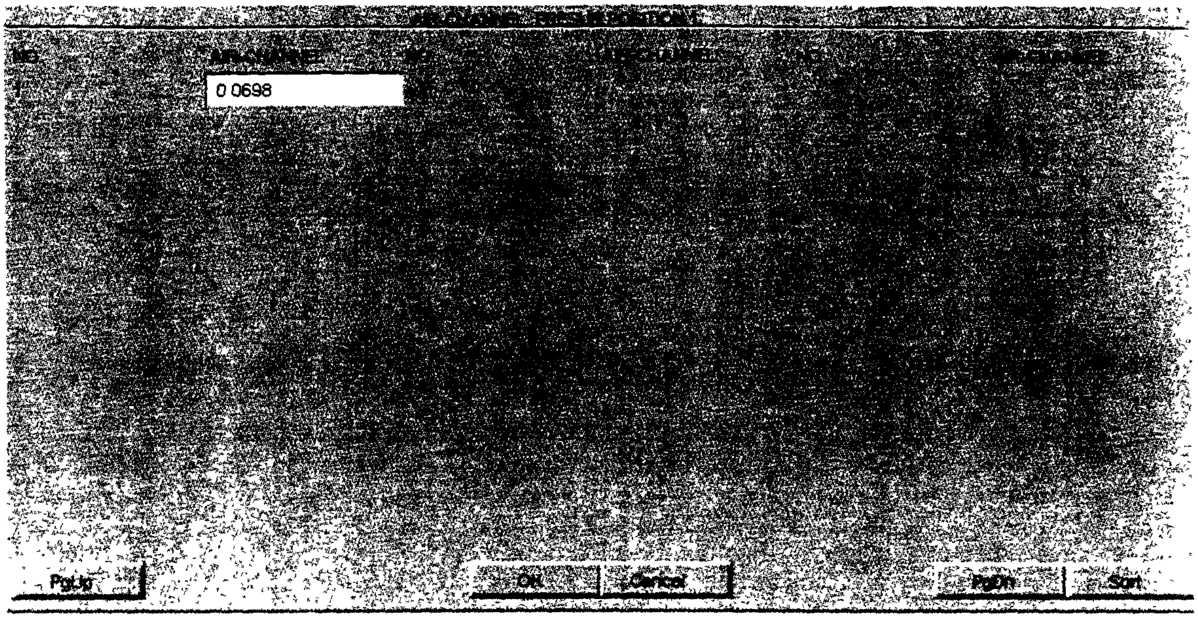


Figure 9.2: Air Channel Create/Modify Submenu

Figure 9.3 shows the air channel data generated.
Click **OK** in Figure 9.3(a), when finished.



(a)



(b)

Figure 9.3: Air Channel Data (a) Geometry and FE Data; (b) Air Pressure Data

At this time, an outline of the blade is drawn on the screen (see Figure 9.4) to enable the selection of the point along the leading edge where the air channel model is to be located. The desired location is at a radial fraction of about 0.7. When a point on the leading edge is selected, the program computes and displays its radial position as shown in Figure 9.4. If the point is far away from radial position **0.7**, then click on **Not Close Enough** to pick another point. If the point is close enough to the desired (as in this case, because the point select had a radial fraction of 0.6987), click on **This is Close Enough** on the menu at the upper right hand side of the blade outline. Then the program will use the data entered earlier to generate the air channel model at this point, with a message displayed to signify that the air channel model has been generated.

Figure 9.4: Blade Outline for Selecting Location of Air-Channel Model

To display the air channel model
Click **Display** on the main menu and then select **Air-Channel FE Model** on the **MODEL DISPLAY** submenu (Figure 9.5)

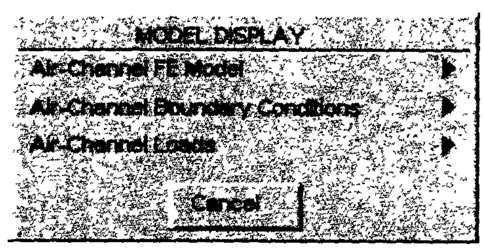


Figure 9.5: Air-Channel Model Display Options

Figure 9.6 shows the finite element mesh of the air-channel model. The boundary conditions can be displayed by clicking on Air-Channel Boundary Conditions on the MODEL DISPLAY submenu (Figure 9.5). The air channel model with the default boundary conditions are shown in Figure 9.6.

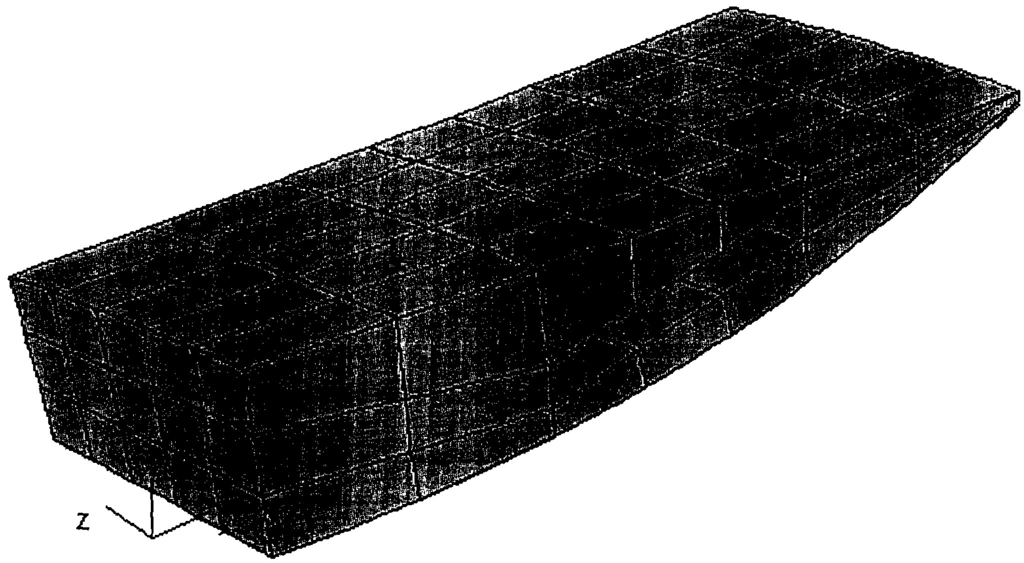


Figure 9.6: Finite Element Mesh of Air-Channel Model

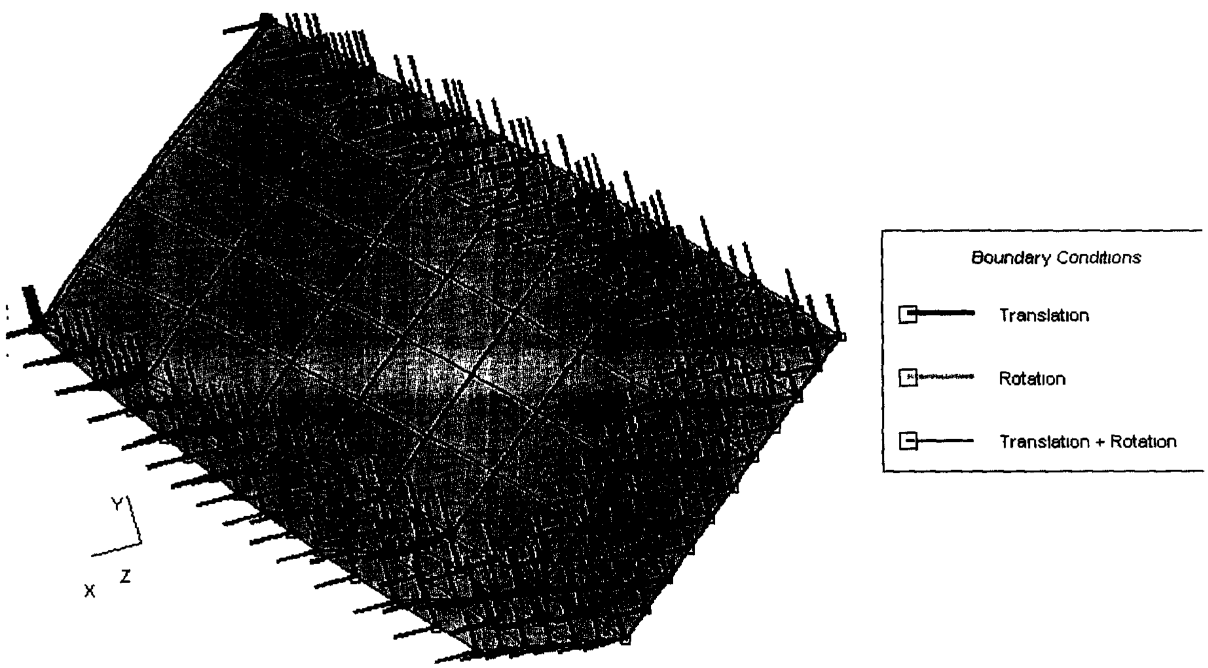


Figure 9.7: Finite Element Mesh of Air-Channel Model Showing Boundary Conditions

9.4 Finite Element Stress Analysis of Air Channel Model

This step involves the performance of the finite element analysis using the VAST finite element code. This is accomplished by the following procedure.

9.4.1 Generate VAST Input File

This step involves generating the input file for a VAST finite element analysis.

Click the **Analysis** on the main menu and select the **Generate VAST Input File** option on the **ANALYSIS** submenu.

During the air-channel model generation, the program generates a file named PREFIX.CHL, which is a complete VAST input deck. Therefore, in this step, all that is done is to copy the PREFIX.CHL file into another file called PREFIXC.USE, which is used for the VAST analysis. This way, any existing VAST files with the original PREFIX name are maintained and not over written. It should be noted that the air-channel data is still retained in the original prefix name.

9.4.2 Finite Element Stress Analysis

The process of performing the finite element stress analysis is similar to the one described in Section 3.3.2.

Click **Analysis** on the main menu, then click **Start VAST Analysis** on the **Analysis** submenu (see Figure 3.10). When the VAST analysis status screen is shown (Figure 3.11) click **Start** to start the VAST stress analysis. Click **OK** when the analysis is complete and return to the PVAST main menu.

9.5 Post- Processing Stress Results

Only the general-purpose plotting options are available for the air-channel model. The procedure for displaying results is similar to that described in Section 5.3.2. Following that procedure, displacement and stress responses from the air-channel model analysis have been plotted. Figure 9.7 (a) shows the deformed shape superimposed on the undeformed air-channel configuration, and Figure 9.7(b) shows contours of the model displacements. Contours of the von-Mises stresses are plotted in Figure 9.8 (a) and (b) using the contour fringe and line plotting options, respectively.

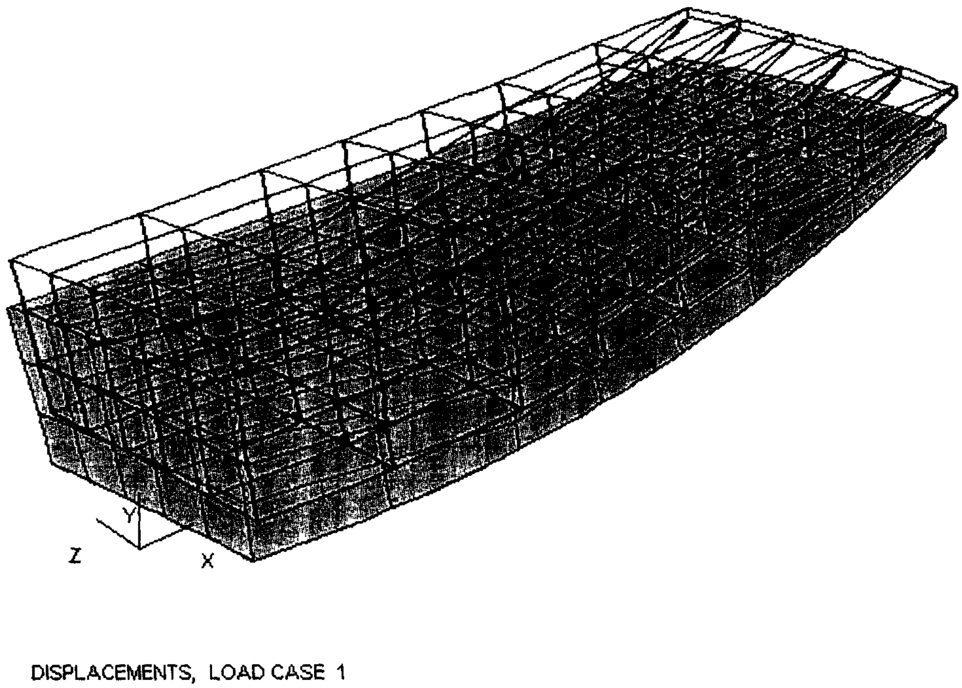


Figure 9.8

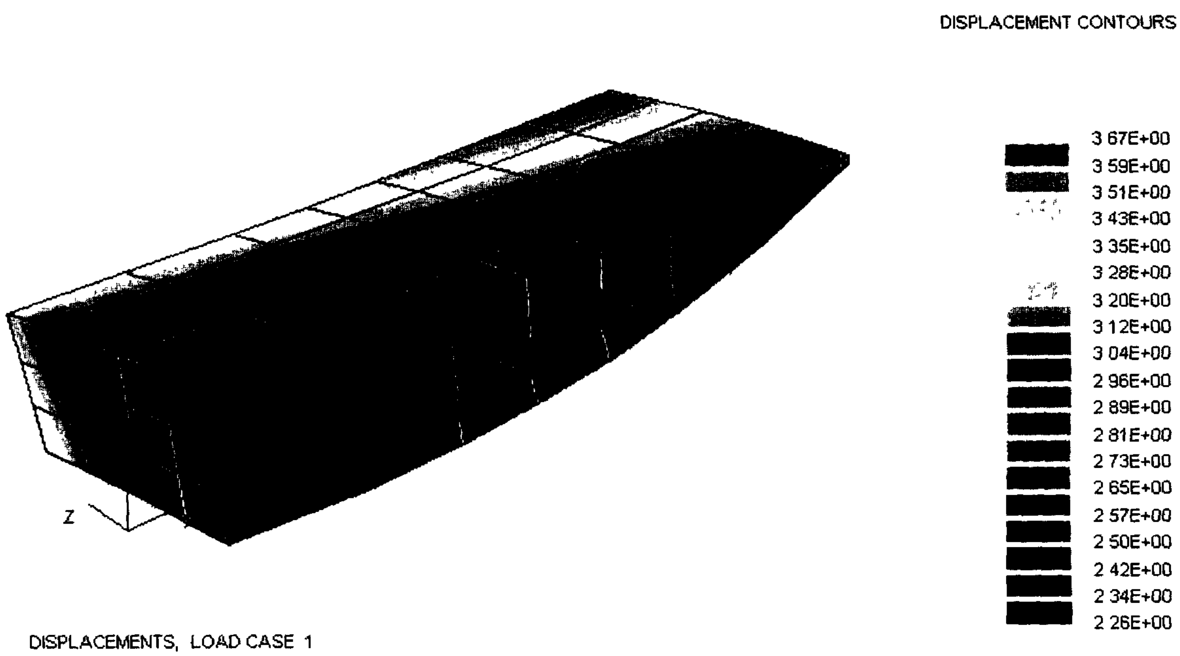
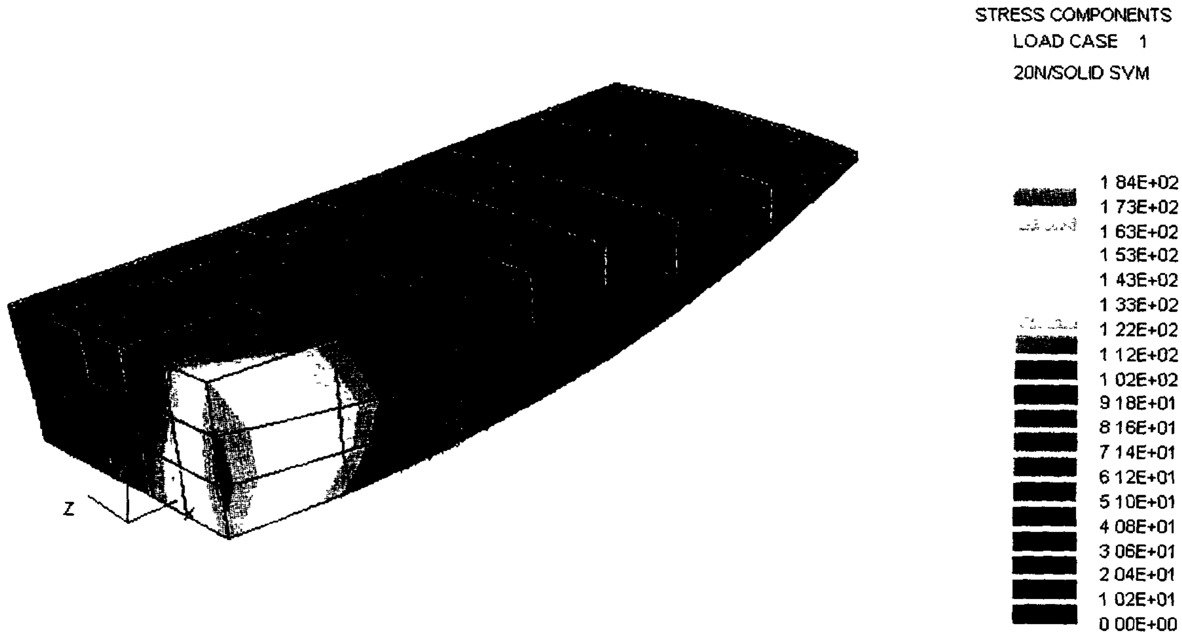
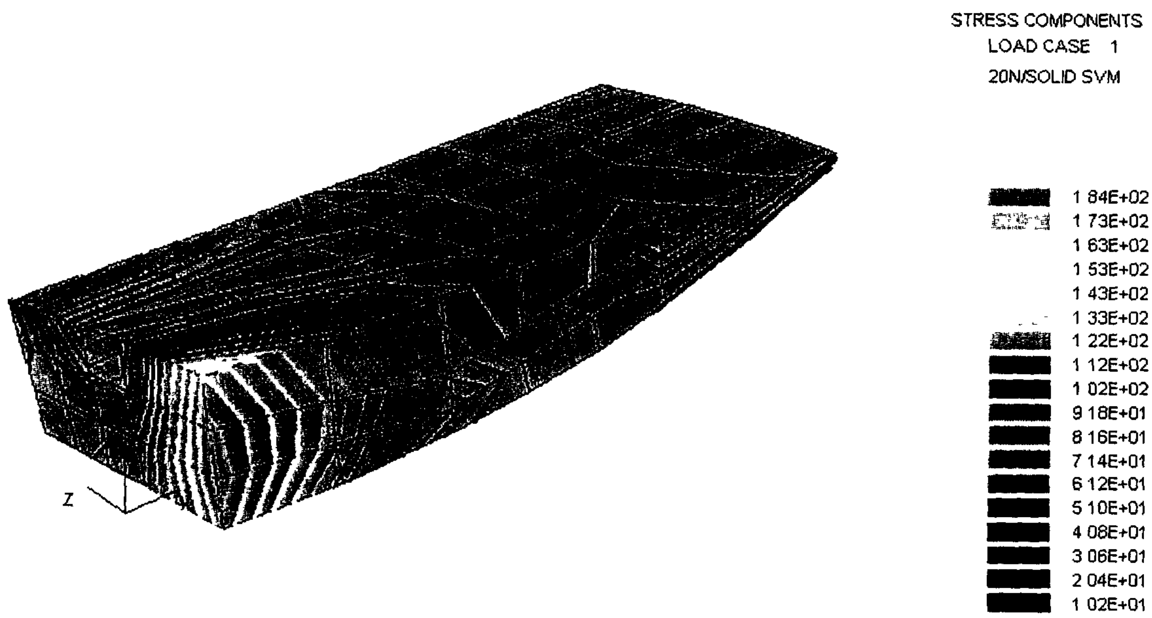


Figure 9.9: Air-Channel Displacement Responses (a) Deformed Shape Superimposed on Undeformed Configuration; (b) Displacement Contours



(a)



(b)

Figure 9.10: Air-Channel von-Mises Stresses (a) Fringe Plot; (b) Contour Line Plot

APPENDIX A:

BLADE GEOMETRY FILE FORMATS AND SAMPLE FILES

1. A.1 Format of PREFIX.DAT File

Headers are used to distinguish various sections of the blade geometry data.

2. A. Basic Data

Header = BASIC

Card 1 (A40)
TITLE = 40 character title

Card 2 (A$)
UNIT = 4 character description of length units
— IN. for length in inches
— MM. for length in millimeters

Card 3 (F10.3)
RR = full radius

Card 4 (I5)
IHAND = parameter for describing the divertion of rotation of the propeller
— (1) for right hand rotation (clockwise looking forward)
— (-1) for left hand rotation (anticlockwise looking forward)

Card 5 (I5)
K = number of radial sections

Card 6 (A1)
PITCH = pitch type
— A for pitch defined as angles
— P for pitch defined as distance travelled in one rotation
— D for pitch as pitch diameter ratio

Card 6 (A1)
SKEW = skew type
— A for skew defined as angle
— D for skew defined as linear measure

Card 7 (I5)
1FORMAT = section data format
— 1 for non-dimensional ratios of maximum thickness
— 2 for NACA section data

— 3 for section offsets

Card 8 (I5)
L = number of chord stations

Card 9 (I5)
1CHORD = — 1 same chord fractions used throughout
— 2 different chord fractions are used

B. Radial Data

Header = RADIAL

Repeat Card 1 for K Sections

Card 1 (F10.3)
R = section radius as fraction of full radius.

Repeat Card 2 for K Sections

Card 2 (F10.3)
C = chord length

Repeat Card 3 for K Sections

Card 3 (F10.3)
P = pitch

Repeat Card 4 for K Sections

Card 4 (F10.3)
S = skew

Repeat Card 5 for K Sections

Card 5 (F10.3)
RK = rake

C. Chordwise Data

Header = CHORDWISE

Repeat Card 1 for K Radial Stations

Card 1 (F10.3)
MAXTHK = maximum thickness/chord ratio

Repeat Card 2 for K Radial Stations

Card 2 (F10.3)
MAXCAM = maximum camber/chord ratio

Repeat Card 3 for L Chord Stations

Card 3 (F10.3)
PERCHORD= chord station as percentage of the chord length

Repeat Card 4 for L Chord Stations

Card 4 (F10.3)
HLFTHK = half thickness as percentage of chord length

Repeat Card 5 for L Chord Stations

Card 5 (F10.3)
CAMB = camber offset as percentage of chord length

Card 6 (I5)
IMOD = modification option
— O for no modification of chordwise data
— I for modification of chord data

Card 7 (I5)
NCHORD = number of existing chord stations

Card 8 (I6)
NCOS = number of COS function generated stations

D. Edge Data

Header = EDGES

Card 1 (I5)
NLEP = number of additional leading edge points

Card 2 (I5)

NTEP = number of additional trailing edge points

Card 3 (I5)
IELLIP = type of edge generated
— 1 for elliptical
— 2 for radial

E. Base Data

Header = BASE

Card 1 (I5)
IBASE = base type
— 1 for hub
— 2 for palm

Card 2 (I5)
NDIAM = number of hub diameters

Repeat Card 3 for NDIAM Diameters

Card 3 (2F10.3)
DIA = diameter
LOC = location at which diameter is measured along the shaft axis

Card 4 (F10.3)
FDIA = forward diameter to origin

A.2 FORMAT OF PREFIX.T16 FILE

Card 1 (A40)
TITLE = 40 character title

Card 2 (I5)
ICHORD = code for specifying chordwise positions for defining face and back offsets. If =1, chordwise positions for the first section are provided, and those for the remaining sections are assumed to be identical. If =2, chordwise positions are provided for each section.

Card 3 (2I5)
K = number of radial positions where data are provided (3=K=43)

L = number of chordwise positions where data are provided (3=L=41)

Card 4 (F10.4)
RR = propeller full radius (length)

Card 5 (see Figure A-1) (SE10.3)
R section radial location as fraction of full radius

P = pitch angle (degrees), positive counter clockwise

S = section chord length

SK = skew (length), positive for section shifted in direction of trailing edge

RK = rake (length), positive aft

Card 6 (C_{1j}, j=1,L) (8F10.4)
$C_{(1,1)}$ = the chord fractions for the face and back surface offsets for the first section (see Figure A.1)
?
$C_{(1,L)}$

Repeat Card 7 for i=2,K

Card 7 (C_{ij}, j=1,L) (Omit if ICHORD=1) (8F10.4)
$C_{i,1}$ = the chord fractions for the face and back surface offsets for sections 2 to K
?
$C_{i,,L}$

Repeat Cards 8 and 9 for j=1,L

Card 8 (TB_{ji}, i=1,K) (8F10.4)

$TB_{(j,1)}$ = back surface offsets (length), i.e. the z' coordinate in the
?   developed sections (see Figure A.1) (negative value for -z'
$TB_{(j,K)}$ coordinate)

Card 9 (TF_{ji}, i=1,K) (8F10.4)

$TF_{(j,1)}$ = face surface offsets
?
$TF_{j,K)}$

A.3 NRC45.DAT Sample File

```
BASIC
QUEST PROPELLER BLADE #45 REF.DWG.CFHQ.#
IN.
    60.000  FULL RADIUS
    1   HAND
   11  NO OF SECTIONS
A
D
    2  SECTION DATA FORMAT
   17   NUMBER OF CHORD STATIONS
    1 SAME FRACTIONS THROUGHOUT
RADIAL
      .250  R
      .300  R
      .400  R
      .500  R
      .600  R
      .700  R
      .800  R
      .900  R
      .950  R
      .975  R
     1.000  R
    18.000  CHORD
    17.560  CHORD
    17.240  CHORD
    17.840  CHORD
    18.500  CHORD
    18.720  CHORD
    17.940  CHORD
    15.300  CHORD
    12.860  CHORD
    10.840  CHORD
     5.300  CHORD
    57.000  P ANGLE
    52.900  P ANGLE
    46.930  P ANGLE
    42.400  P ANGLE
    37.980  P ANGLE
    33.450  P ANGLE
    28.630  P ANGLE
    24.120  P ANGLE
    22.370  P ANGLE
    21.650  P ANGLE
    21.050  P ANGLE
     1.638  SKEW
      .285  SKEW
    -1.030  SKEW
    -1.602  SKEW
    -1.781  SKEW
    -1.624  SKEW
    -1.198  SKEW
     -.607  SKEW
     -.256  SKEW
     -.069  SKEW
      .000  SKEW
      .658  RAKE
     2.711  RAKE
```

```
     5.517  RAKE
     7.757  RAKE
     9.675  RAKE
    11.382  RAKE
    12.872  RAKE
    14.312  RAKE
    15.007  RAKE
    15.368  RAKE
    15.728  RAKE
CHORDWISE
      .256  MAX TH/CHORD
      .239  MAX TH/CHORD
      .207  MAX TH/CHORD
      .167  MAX TH/CHORD
      .127  MAX TH/CHORD
      .092  MAX TH/CHORD
      .065  MAX TH/CHORD
      .049  MAX TH/CHORD
      .050  MAX TH/CHORD
      .057  MAX TH/CHORD
      .113  MAX TH/CHORD
      .025  MAX CAM/CHORD
      .046  MAX CAM/CHORD
      .068  MAX CAM/CHORD
      .072  MAX CAM/CHORD
      .065  MAX CAM/CHORD
      .052  MAX CAM/CHORD
      .033  MAX CAM/CHORD
      .015  MAX CAM/CHORD
      .006  MAX CAM/CHORD
      .002  MAX CAM/CHORD
     -.006  MAX CAM/CHORD
      .000  % CHORD
     1.250  % CHORD
     2.500  % CHORD
     5.000  % CHORD
     7.500  % CHORD
    10.000  % CHORD
    15.000  % CHORD
    20.000  % CHORD
    30.000  % CHORD
    40.000  % CHORD
    50.000  % CHORD
    60.000  % CHORD
    70.000  % CHORD
    80.000  % CHORD
    90.000  % CHORD
    95.000  % CHORD
   100.000  % CHORD
      .000  HALF THICK % CHORD
     1.292  HALF THICK % CHORD
     1.805  HALF THICK % CHORD
     2.509  HALF THICK % CHORD
     3.032  HALF THICK % CHORD
     3.457  HALF THICK % CHORD
     4.135  HALF THICK % CHORD
     4.664  HALF THICK % CHORD
     5.417  HALF THICK % CHORD
     5.885  HALF THICK % CHORD
     6.000  HALF THICK % CHORD
```

```
      5.835  HALF THICK % CHORD
      5.269  HALF THICK % CHORD
      4.199  HALF THICK % CHORD
      2.517  HALF THICK % CHORD
      1.415  HALF THICK % CHORD
       .120  HALF THICK % CHORD
       .000  CAMBER OFFSET % CHORD
       .296  CAMBER OFFSET % CHORD
       .585  CAMBER OFFSET % CHORD
      1.140  CAMBER OFFSET % CHORD
      1.665  CAMBER OFFSET % CHORD
      2.160  CAMBER OFFSET % CHORD
      3.060  CAMBER OFFSET % CHORD
      3.840  CAMBER OFFSET % CHORD
      5.040  CAMBER OFFSET % CHORD
      5.760  CAMBER OFFSET % CHORD
      6.000  CAMBER OFFSET % CHORD
      5.760  CAMBER OFFSET % CHORD
      5.040  CAMBER OFFSET % CHORD
      3.840  CAMBER OFFSET % CHORD
      2.160  CAMBER OFFSET % CHORD
      1.140  CAMBER OFFSET % CHORD
       .000  CAMBER OFFSET % CHORD
     0 MODIFICATION OPTION
    17 NO OF EXISTING CHORD STATIONS
     0 NO. OF COS FUNCTION GENERATED STATIONS
EDGES
     3 NO. OF ADDITIONAL LEADING EDGE PTS.
     0 NO. OF ADDITIONAL TRAILING EDGE PTS.
     1 ELLIPTICAL TYPE LEADING EDGE
     0  NO. OF ADDITIONAL TIP RADII
       .000 BLADE TIP PITCH
       .000  BLADE TIP SKEW
       .000  BLADE TIP RAKE
       .000  BLADE TIP POINT OF MAX THICK.
     0       TIP GENERATION BY RADIAL METHOD
       .000  RADIUS OF TIP PROFILE
     0  DISTANCE NOT KNOWN FROM REF TO CENTER OF BLADE TIP
       .000  DIST FROM REF AXIS TO CENTER OF BLADE TIP
     0  COORINATE DATA PRINT OPTION
       .000  ROTATION ANGLE FOR BLADE SURFACE MEASUREMENT
       .000  SHIM THICKNESS FOR BLADE MEASUREMENT
    .5500   .4900   .3600   .2400   .1500   .0800   .0630   .0630  LE
    .0630   .0630   .0630
    .0000   .0000   .0000   .0000   .0000   .0000   .0000   .0000  LE
    .0000   .0000   .0000
       .000       .000        .000        .000        .000        .000        .000
.000  ANTISING ANG.
       .000      .000      .000
       .000       .000        .000        .000        .000        .000        .000
.000  ANTISING LENGTH
       .000      .000      .000
BASE
     1  BLADE BASE TYPE
     3  NUMBER OF HUB DIAMETERS
     26.500      .000  DIA. AND LOCATION
     24.800    12.750  DIA. AND LOCATION
     19.000    28.125  DIA. AND LOCATION
     12.750FWD. DIA. TO ORIGIN
```

A.4 P5363.DAT Sample File

```
BASIC
QUEST SKEWED PROPELLER NO.5363 DWG.5561-6
MM.
  1524.000  FULL RADIUS
     1   HAND
    12  NO OF SECTIONS
     3  SECTION DATA FORMAT
    13  NUMBER OF CHORD STATIONS
     1  SAME FRACTIONS THROUGHOUT
     2  PMT IS CHORD DATUM
RADIAL
      0.200  R
      0.250  R
      0.300  R
      0.400  R
      0.500  R
      0.600  R
      0.700  R
      0.800  R
      0.850  R
      0.900  R
      0.950  R
      0.975  R
    475.000  CHORD
    491.000  CHORD
    507.000  CHORD
    535.000  CHORD
    558.000  CHORD
    581.000  CHORD
    605.000  CHORD
    615.000  CHORD
    598.000  CHORD
    560.000  CHORD
    477.000  CHORD
    370.000  CHORD
A
     60.681 P ANGLE
     56.647 P ANGLE
     53.157 P ANGLE
     47.398 P ANGLE
     42.401 P ANGLE
     37.553 P ANGLE
     32.359 P ANGLE
     26.862 P ANGLE
     24.440 P ANGLE
     22.462 P ANGLE
     20.879 P ANGLE
     20.200 P ANGLE
D
   -200.000  SKEW
   -173.000  SKEW
   -139.000  SKEW
    -59.000  SKEW
     45.000  SKEW
    171.000  SKEW
    337.000  SKEW
    542.000  SKEW
    670.000  SKEW
```

```
   808.000  SKEW
   967.000  SKEW
  1059.000  SKEW
   -53.744  RAKE
   -67.181  RAKE
   -80.617  RAKE
  -107.489  RAKE
  -134.361  RAKE
  -161.233  RAKE
  -188.106  RAKE
  -214.978  RAKE
  -228.414  RAKE
  -241.850  RAKE
  -255.286  RAKE
  -262.004  RAKE
CHORDWISE
  0.4968  0.4929  0.4892  0.4692  0.4301  0.3769  0.3355  0.3073 PMT
  0.2993  0.3000  0.2998  0.3000
  1.0000  0.9500  0.9000  0.8000  0.6000  0.4000  0.2000  0.0000 CHD FRC
CHD.FRC
  0.2000  0.4000  0.6000  0.8000  1.0000
   -52.300   -69.700   -77.000   -87.600  -102.900  -112.800  -117.900TB
TB
  -119.600  -115.500  -105.700   -91.500   -73.800   -46.400
   -45.100   -62.000   -69.200   -79.800   -95.300  -105.100  -110.400TB
TB
  -112.000  -108.200   -98.500   -84.200   -66.500   -38.800
   -37.700   -54.100   -61.200   -71.900   -87.400   -97.200  -102.700TB
TB
  -104.300  -100.800   -91.300   -77.300   -59.600   -31.700
   -23.300   -38.600   -45.800   -56.400   -72.100   -82.100   -87.700TB
TB
   -89.300   -86.300   -78.000   -64.800   -47.500   -19.600
   -11.400   -25.100   -32.000   -42.400   -57.600   -67.500   -73.000TB
TB
   -74.600   -72.200   -65.000   -53.200   -36.900   -10.300
    -3.600   -15.400   -21.600   -31.300   -45.200   -54.300   -59.500TB
TB
   -61.000   -59.000   -52.800   -42.500   -27.700    -3.600
    -0.800   -10.300   -15.400   -23.600   -35.400   -43.100   -47.600TB
TB
   -48.900   -47.200   -42.100   -33.400   -21.000    -0.800
    -2.300    -9.500   -13.300   -19.400   -28.300   -34.100   -37.300TB
TB
   -38.300   -37.000   -33.300   -26.800   -17.700    -2.300
    -4.100    -9.900   -13.100   -18.100   -25.400   -30.100   -32.800TB
TB
   -33.500   -32.500   -29.400   -24.200   -16.900    -4.100
    -6.300   -11.200   -13.500   -17.500   -23.100   -26.600   -28.600TB
TB
   -29.200   -28.400   -26.100   -22.100   -16.600    -6.300
    -8.700   -12.600   -14.300   -17.200   -21.200   -23.700   -25.000TB
TB
   -25.400   -24.800   -23.100   -20.400   -16.800    -8.700
    -9.900   -13.300   -14.600   -17.000   -20.300   -22.300   -23.400TB
TB
   -23.700   -23.200   -21.900   -19.700   -16.900    -9.900
   -52.300   -35.900   -29.700   -21.200   -10.200    -4.000    -0.900TF
TF
     0.000    -1.700    -6.100   -13.300   -24.500   -46.400
```

```
   -45.100   -29.700   -24.200   -16.900    -7.800    -3.000    -0.600TF
TF
     0.000    -1.100    -4.200    -9.900   -19.100   -38.800
   -37.700   -23.600   -18.700   -12.700    -5.400    -1.900    -0.300TF
TF
     0.000    -0.700    -2.800    -7.200   -14.400   -31.700
   -23.300   -11.900    -8.500    -4.800    -1.200    -0.100     0.200TF
TF
     0.000    -0.200    -1.100    -3.300    -7.200   -19.600
   -11.400    -2.500    -0.500     1.200     1.900     1.200     0.500TF
TF
     0.000     0.000    -0.200    -0.900    -2.300   -10.300
    -3.600     3.000     4.200     4.600     3.500     1.900     0.600TF
TF
     0.000     0.100     0.500     0.800     1.200    -3.600
    -0.800     4.100     4.900     5.000     3.600     1.900     0.600TF
TF
     0.000     0.200     0.800     1.600     2.500    -0.800
    -2.300     1.600     2.300     2.800     2.100     1.100     0.400TF
TF
     0.000     0.100     0.400     0.700     1.100    -2.300
    -4.100    -0.600     0.100     0.900     1.000     0.600     0.200TF
TF
     0.000     0.000     0.000    -0.200     0.000    -4.100
    -6.300    -3.100    -2.200    -1.100    -0.200     0.100     0.100TF
TF
     0.000    -0.100    -0.500    -1.200    -1.900    -6.300
    -8.700    -5.600    -4.700    -3.200    -1.400    -0.500    -0.100TF
TF
     0.000    -0.200    -0.900    -2.100    -3.500    -8.700
    -9.900    -6.800    -5.800    -4.200    -2.000    -0.700    -0.100TF
TF
     0.000    -0.300    -1.200    -2.600    -4.200    -9.900
EDGES
    5 NO. OF ADDITIONAL LEADING EDGE PTS.
    5 NO. OF ADDITIONAL TRAILING EDGE PTS.
    1  ELLIPTICAL TYPE LEADING EDGE
    3  NO. OF ADDITIONAL BLADE TIP RADII
    19.607 BLADE TIP PITCH ANGLE
  1168.000  BLADE TIP SKEW
  -270.000  BLADE TIP RAKE
     0.300  BLADE TIP PT OF MAX THK
     0      TIP GENERATION BY RADIAL METHOD
    10.000  RADIUS OF TIP PROFILE
     1      DISTANCE KNOWN FROM REF TO CENTER OF BLADE TIP
      .000  DISTANCE FROM REF AXIS TO CENTER OF BLADE TIP
     3      COORDINATE DATA PRINT OPTION
      .000  ROTATION ANGLE FOR BLADE SURFACE MEASUREMENT
      .000  SHIM THICKNESS FOR DREA MEAS MACHINE
  7.0000  6.9000  6.7500  6.5000  6.0000  3.0000  2.5000  2.2500 LE
  2.0000  2.0000  2.0000  2.0000
  7.0000  6.9000  6.7500  6.5000  6.0000  5.5000  5.0000  4.5000 1E
  4.7500  4.0000  4.0000  4.0000
BASE
    0  BLADE BASE TYPE
```

A.5 P4388.DAT Sample File

```
BASIC
 HIGHLY SKEWED PROPELLER NO.4388
IN.
     6.000  FULL RADIUS
    1   HAND
    9  NO OF SECTIONS
A
D
    3  SECTION DATA FORMAT
   17   NUMBER OF CHORD STATIONS
    1 SAME FRACTIONS THROUGHOUT
    1 50% CHORD DATUM
RADIAL
      .200  R
      .300  R
      .400  R
      .500  R
      .600  R
      .700  R
      .800  R
      .900  R
      .950  R
     3.840  CHORD
     4.362  CHORD
     4.858  CHORD
     5.270  CHORD
     5.532  CHORD
     5.546  CHORD
     5.216  CHORD
     4.336  CHORD
     3.330  CHORD
    55.813  P ANGLE
    44.664  P ANGLE
    37.214  P ANGLE
    32.237  P ANGLE
    28.626  P ANGLE
    25.967  P ANGLE
    24.029  P ANGLE
    22.662  P ANGLE
    22.069  P ANGLE
      .000  SKEW
      .471  SKEW
     1.256  SKEW
     2.356  SKEW
     3.770  SKEW
     5.498  SKEW
     7.540  SKEW
     9.896  SKEW
    11.090  SKEW
      .000  RAKE
      .000  RAKE
      .000  RAKE
      .000  RAKE
      .000  RAKE
      .000  RAKE
      .000  RAKE
      .000  RAKE
      .000  RAKE
```

```
CHORDWISE
 .0000   .0125   .0250   .0500   .0750   .1000   .2000   .3000 CHD.FRC.
 .4000   .4500   .5000   .6000   .7000   .8000   .9000   .9500 CHD.FRC.
1.0000
     .000     -.054     -.074     -.106     -.132     -.154     -.219 TB
    -.258     -.279     -.284     -.283     -.269     -.237     -.188 TB
    -.105     -.060     -.013
     .000     -.054     -.078     -.116     -.147     -.174     -.252 TB
    -.300     -.328     -.334     -.334     -.320     -.285     -.222 TB
    -.123     -.067     -.011
     .000     -.050     -.074     -.111     -.142     -.168     -.246 TB
    -.295     -.323     -.329     -.331     -.318     -.283     -.222 TB
    -.121     -.065     -.010
     .000     -.045     -.067     -.102     -.131     -.156     -.230 TB
    -.274     -.303     -.310     -.311     -.299     -.268     -.210 TB
    -.114     -.060     -.008
     .000     -.040     -.060     -.092     -.119     -.142     -.210 TB
    -.254     -.278     -.285     -.286     -.276     -.247     -.194 TB
    -.104     -.055     -.006
     .000     -.034     -.052     -.081     -.105     -.126     -.188 TB
    -.228     -.251     -.256     -.258     -.250     -.224     -.176 TB
    -.094     -.048     -.005
     .000     -.028     -.043     -.068     -.088     -.106     -.159 TB
    -.193     -.213     -.218     -.219     -.212     -.191     -.150 TB
    -.080     -.040     -.004
     .000     -.020     -.030     -.048     -.063     -.076     -.114 TB
    -.139     -.153     -.157     -.158     -.153     -.138     -.108 TB
    -.058     -.029     -.002
     .000     -.014     -.022     -.034     -.045     -.054     -.084 TB
    -.099     -.109     -.112     -.113     -.109     -.098     -.077 TB
    -.041     -.021     -.002
     .000      .038      .046      .059      .068      .076      .097 TF
     .107      .111      .111      .108      .098      .081      .061 TF
     .042      .030      .013
     .000      .024      .025      .026      .025      .024      .019 TF
     .013      .008      .005      .002      .005      .011      .012 TF
     .004      .010      .011
     .000      .016      .014      .009      .004     -.001     -.017 TF
    -.030     -.039     -.043     -.046     -.051     -.052     -.043 TF
    -.014      .001      .010
     .000      .010      .006     -.003     -.010     -.017     -.040 TF
    -.054     -.068     -.072     -.075     -.078     -.076     -.062 TF
    -.025     -.006      .008
     .000      .005     -.001     -.011     -.021     -.030     -.056 TF
    -.076     -.088     -.092     -.095     -.097     -.092     -.074 TF
    -.032     -.011      .006
     .000      .000     -.006     -.018     -.029     -.038     -.068 TF
    -.089     -.102     -.106     -.109     -.110     -.102     -.082 TF
    -.038     -.014      .005
     .000     -.002     -.010     -.022      .000     -.042     -.071 TF
    -.092     -.104     -.108     -.110     -.110     -.102     -.082 TF
    -.039     -.016      .004
     .000     -.003     -.009     -.018     -.026     -.034     -.057 TF
    -.072     -.082     -.085     -.087     -.086     -.080     -.064 TF
    -.030     -.012      .002
     .000     -.002     -.006     -.012     -.018     -.023     -.036 TF
    -.049     -.056     -.058     -.059     -.059     -.055     -.044 TF
    -.021     -.009      .002
EDGES
    5 NO. OF ADDITIONAL LEADING EDGE PTS.
```

```
    0 NO. OF ADDITIONAL TRAILING EDGE PTS.
    1 ELLIPTICAL TYPE LEADING EDGE
    3  NO. OF ADDITIONAL TIP RADII
    21.591 BLADE TIP PITCH
    12.564  BLADE TIP SKEW
      .000  BLADE TIP RAKE
      .000  BLADE TIP POINT OF MAX THICK.
    1       ELLIPTICAL TIP GENERATION
      .040  SEMITHICKNESS OF ELLIPTICAL PROFILE
    1  DISTANCE KNOWN FROM REF TO CENTER OF BLADE TIP
     -.040  DIST FROM REF AXIS TO CENTER OF BLADE TIP
    1  COORINATE DATA PRINT OPTION
      .000  ROTATION ANGLE FOR BLADE SURFACE MEASUREMENT
      .000  SHIM THICKNESS FOR BLADE MEASUREMENT
   .0260   .0168   .0108   .0068   .0044   .0028   .0028   .0028  LE
   .0028
   .0000   .0000   .0000   .0000   .0000   .0000   .0000   .0000  LE
   .0000
      .000       .000       .000       .000       .000       .000       .000
.000  ANTISING ANG.
      .000
      .000       .000       .000       .000       .000       .000       .000
.000  ANTISING LENGTH
      .000
BASE
    0  BLADE BASE TYPE
```

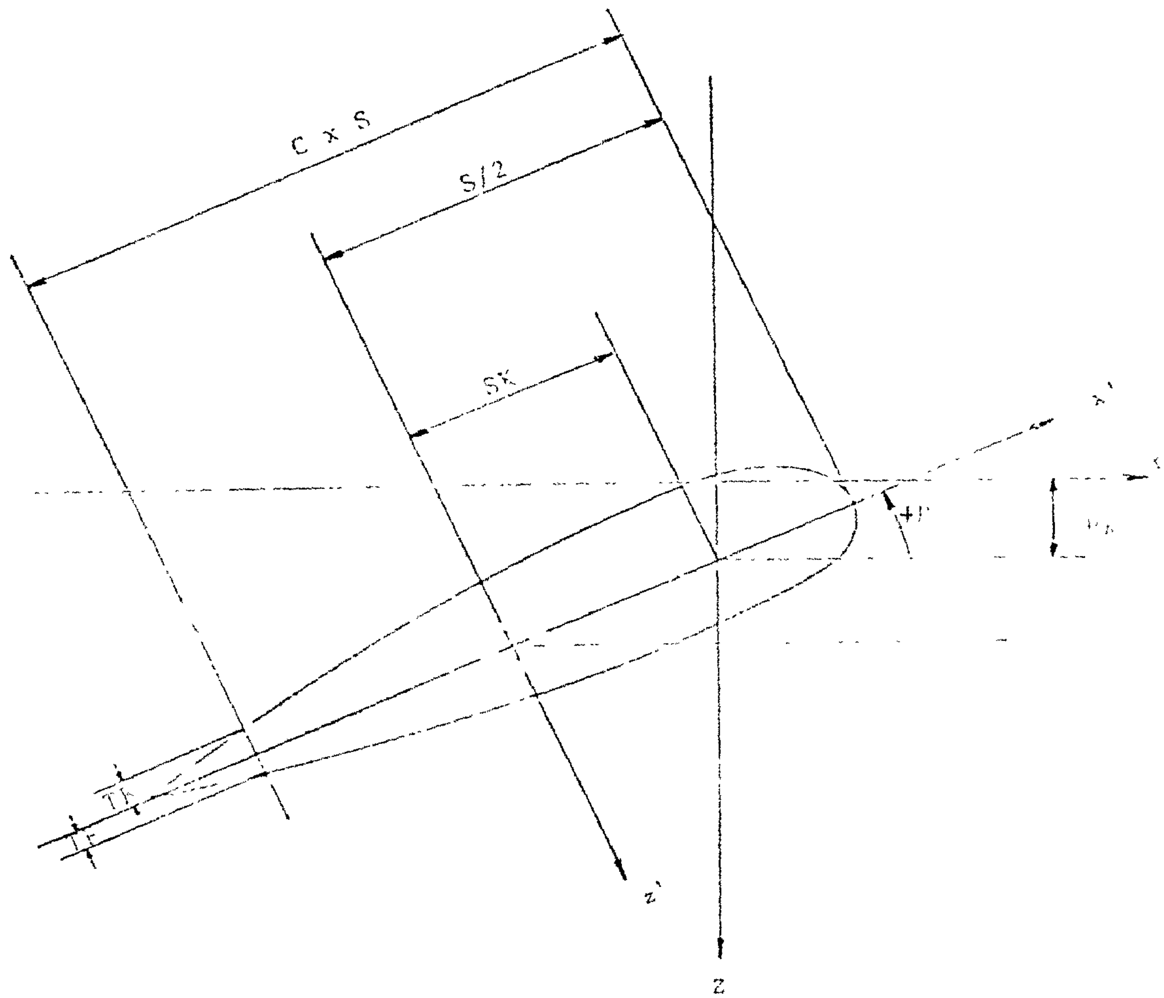


Figure A.1: Diagramatic Explanation of Parameters Used to Describe Blade Section Geometry

APPENDIX B:

P5363.LIN SAMPLE FILE

P5363.LIN Sample File

```
1     0 NSMB   5363         MODEL OF QUEST NEW DESIGN PROPELLER      -
    2    1    0
   12   19
    0.2366     0.2782     0.3338     0.4011     0.4771     0.5584     0.6416
0.7229
    0.7989    0.8662    0.9218    0.9634
    0.0000     0.0085     0.0287     0.0618     0.1068     0.1625     0.2275
0.2998
    0.3777     0.4589     0.5411     0.6223     0.7002     0.7725     0.8375
0.8932
    0.9382    0.9713    1.0000
    0.1714     0.1427     0.0650     0.0827     0.0577     0.0572     0.0525
0.0536
    0.0583     0.0581     0.0572     0.0570     0.0623     0.0617     0.0641
0.0642
    0.0929    0.1080    0.1714
    0.1714     0.1489     0.0566     0.0812     0.0599     0.0637     0.0639
0.0684
    0.0759     0.0798     0.0825     0.0831     0.0874     0.0844     0.0840
0.0813
    0.1093    0.1226    0.1714
    0.1804     0.1460     0.0619     0.0781     0.0508     0.0489     0.0434
0.0439
    0.0483     0.0479     0.0474     0.0483     0.0556     0.0568     0.0617
0.0645
    0.0964    0.1131    0.1804
    0.1804     0.1480     0.0565     0.0802     0.0589     0.0628     0.0630
0.0683
    0.0768     0.0814     0.0844     0.0853     0.0896     0.0869     0.0874
0.0857
    0.1143    0.1275    0.1804
    0.1947     0.1573     0.0541     0.0710     0.0424     0.0398     0.0337
0.0337
    0.0380     0.0379     0.0382     0.0403     0.0491     0.0515     0.0577
0.0617
    0.0960    0.1146    0.1947
    0.1947     0.1571     0.0612     0.0839     0.0628     0.0667     0.0670
0.0724
    0.0811     0.0857     0.0887     0.0897     0.0944     0.0923     0.0932
0.0913
    0.1195    0.1321    0.1947
    0.2156     0.1633     0.0438     0.0614     0.0325     0.0294     0.0221
0.0220
    0.0268     0.0280     0.0300     0.0333     0.0428     0.0459     0.0529
0.0577
    0.0942    0.1154    0.2156
    0.2156     0.1722     0.0712     0.0925     0.0708     0.0748     0.0755
0.0805
    0.0887     0.0926     0.0949     0.0960     0.1013     0.0996     0.1006
0.0983
    0.1261    0.1378    0.2156
    0.2437     0.1554     0.0324     0.0494     0.0197     0.0158     0.0076
0.0079
    0.0145     0.0185     0.0230     0.0275     0.0377     0.0413     0.0492
0.0549
    0.0934    0.1168    0.2437
    0.2437     0.1937     0.0856     0.1050     0.0832     0.0867     0.0872
0.0917
```

```
   0.0987     0.1012     0.1026     0.1035     0.1093     0.1080     0.1090
0.1063
   0.1330     0.1436     0.2437
   0.2792     0.1413     0.0195     0.0357     0.0041     0.0008     0.0008
0.0008
   0.0041     0.0119     0.0189     0.0246     0.0353     0.0398     0.0488
0.0559
   0.0949     0.1193     0.2792
   0.2792     0.2280     0.1016     0.1194     0.0992     0.1020     0.1018
0.1051
   0.1099     0.1104     0.1108     0.1115     0.1172     0.1160     0.1167
0.1136
   0.1384     0.1477     0.2792
   0.3213     0.1082     0.0066     0.0229     0.0008     0.0008     0.0008
0.0008
   0.0008     0.0103     0.0185     0.0254     0.0369     0.0426     0.0527
0.0607
   0.0987     0.1226     0.3213
   0.3213     0.2646     0.1184     0.1331     0.1146     0.1166     0.1155
0.1172
   0.1191     0.1177     0.1175     0.1178     0.1226     0.1212     0.1213
0.1180
   0.1405     0.1488     0.3213
   0.3682     0.0584     0.0008     0.0119     0.0008     0.0008     0.0008
0.0008
   0.0014     0.0145     0.0229     0.0301     0.0420     0.0483     0.0582
0.0662
   0.1022     0.1250     0.3682
   0.3682     0.3032     0.1322     0.1426     0.1247     0.1265     0.1246
0.1242
   0.1237     0.1210     0.1205     0.1202     0.1237     0.1217     0.1217
0.1190
   0.1400     0.1478     0.3682
   0.4170     0.0008     0.0008     0.0062     0.0008     0.0008     0.0008
0.0008
   0.0110     0.0228     0.0304     0.0368     0.0478     0.0534     0.0623
0.0693
   0.1036     0.1252     0.4170
   0.4170     0.3394     0.1426     0.1474     0.1290     0.1303     0.1273
0.1248
   0.1231     0.1200     0.1190     0.1184     0.1212     0.1190     0.1190
0.1170
   0.1382     0.1470     0.4170
   0.4643     0.0008     0.0008     0.0052     0.0008     0.0008     0.0008
0.0059
   0.0214     0.0317     0.0385     0.0438     0.0535     0.0580     0.0657
0.0711
   0.1053     0.1265     0.4643
   0.4643     0.3733     0.1532     0.1505     0.1301     0.1284     0.1238
0.1206
   0.1188     0.1159     0.1147     0.1140     0.1168     0.1145     0.1146
0.1129
   0.1360     0.1471     0.4643
   0.5064     0.0008     0.0008     0.0028     0.0008     0.0008     0.0069
0.0179
   0.0317     0.0406     0.0469     0.0515     0.0603     0.0632     0.0695
0.0730
   0.1098     0.1321     0.5064
   0.5064     0.4071     0.1623     0.1519     0.1272     0.1225     0.1172
0.1143
```

```
     0.1132     0.1106     0.1092     0.1083     0.1113     0.1090     0.1092
0.1075
     0.1351    0.1495    0.5064
     0.5395     0.0008     0.0008     0.0014     0.0009     0.0071     0.0148
0.0279
     0.0406     0.0474     0.0530     0.0573     0.0653     0.0662     0.0707
0.0715
     0.1062    0.1258    0.5395
     0.5395     0.4226     0.1735     0.1543     0.1273     0.1186     0.1128
0.1096
     0.1091     0.1067     0.1043     0.1021     0.1051     0.1028     0.1032
0.0996
     0.1299    0.1450    0.5395
     0
```

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 EXAMPLES 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)

133

6b NO OF REFS (total cited in document)

6

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 2000-153

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

SECURITY CLASSIFICATION OF FORM
UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM

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 an examples 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. This manual provides a number of example problems, considered with PVAST Version 7.3 for a range of model types, analysis options and results.

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

UNCLASSIFIED

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 DÉFENSE

www.drdc-rddc.dnd.ca