STRAP: Steam Turbine Rotor Analysis Program
Part 1: General Information Manual

NP-1687, Part 1
Research Project 502

Computer Code Manual, January 1981

Prepared by

BATTELLE, COLUMBUS LABORATORIES
505 King Avenue
Columbus, Ohio 43201

SOUTHWEST RESEARCH INSTITUTE
P.O. Drawer 28510
San Antonio, Texas 78284

Compiled by
H. G. Pennick, Southwest Research Institute

Principal Contributors
T. S. Cook, General Electric
J. R. Fox, Battelle, Columbus Laboratories
M. J. Golis, Battelle, Columbus Laboratories
K. E. Pfister, Battelle, Columbus Laboratories
R. J. Warren, Battelle, Columbus Laboratories
C. H. Wells, Southwest Research Institute

Prepared for

Electric Power Research Institute
3412 Hillview Avenue
Palo Alto, California 94304

EPRI Project Manager
F. E. Gelhaus

Reliability, Availability, and Economics Program
Nuclear Power Division

ORDERING INFORMATION

Requests for copies of this report should be directed to Research Reports Center (RRC), Box 50490, Palo Alto, CA 94303, (415) 965-4081. There is no charge for reports requested by EPRI member utilities and affiliates, contributing nonmembers, U.S. utility associations, U.S. government agencies (federal, state, and local), media, and foreign organizations with which EPRI has an information exchange agreement. On request, RRC will send a catalog of EPRI reports.

Prepared by
Battelle, Columbus Laboratories
Columbus, Ohio
and
Southwest Research Institute
San Antonio, Texas

EPRI PERSPECTIVE

PROJECT DESCRIPTION

Reflecting upon the destructive burst of the Gallatin rotor in June 1974, many utility engineers voiced their concerns to EPRI regarding steam turbine boresonic examinations, the analysis associated with those nondestructive examination (NDE) data, and the unexplained engineering judgments which were all part of the manufacturer's recommendation to run or retire the unit. The common message to EPRI from those utility staff members was that they needed a manufacturer-independent means for accumulating and evaluating the bore inspection data so that, from their standpoint, there would be no unknown parameters or "black boxes" involved in the multimillion-dollar run/retire decision. RP502 began in January 1976 with the goal of meeting this need for providing verifying inspection and analysis capability.

This three-part report documents the first generation of computer software for steam turbine rotor analysis (STRAP-1) and, from that standpoint, is a final report. However, since the development of the second generation of improved capability is already underway, these manuals will be augmented and updated.

EPRI reports from the RP502 development efforts are:

- EPRI Interim Report NP-744, Nondestructive Evaluation of Steam Turbine Rotors, April 1978
- EPRI Interim Report NP-923, Steam-Turbine Rotor Reliability--Task Details, November 1978
- EPRI Summary Report NP-923-SY, Reliability of Steam Turbine Rotors, October 1978
- EPRI NP-923 Supplemental Report, Effects of Temper Embrittlement on the Fracture and Mechanical Properties of an Air Cast CrMoV Steam Turbine Rotor Material, May 1979
- The final report for RP502 is scheduled for publication in June 1981

Related reports that focus on materials evaluations are:

- EPRI Technical Report NP-1023, Fracture and Fatigue Properties of 1Cr-Mo-V Bainitic Turbine Rotor Steels (RP700-1), March 1979
- EPRI Final Report NP-1501, Elimination of Impurity-Induced Embrittlement in Steel, Part 1: Impurity Segregation and Temper Embrittlement, Part 2: High-Temperature Cracking--Stress-Relief and Creep Cracking (RP559), September 1980

PROJECT OBJECTIVES

The primary goal of this project is to develop a computerized rotor-lifetime-prediction system to assist utilities in performing run/retire analyses. These analyses can be completed by using actual steam turbine operational and inspection data or can be done on a parametric basis to further understand the sensitivity of the run/retire conclusion to key mechanistic assumptions and model approximations. A second major pursuit of this research has been (1) to evaluate the state-of-the-art of the nondestructive inspection systems and processes which are used in commercial rotor bore examinations and (2) to design, fabricate, and test improved and completely new inspection systems and concepts.

PROJECT RESULTS

The best and perhaps the only legitimate way of judging the success of EPRI-sponsored research is to evaluate the degree of interest in and use of project results by U.S. utilities. The high degree of interaction in RP502 by numerous utility staff has been an exciting and rewarding part of the research. The role played by the American Electric Power (AEP) Service Corporation as our host utility plus their independent but coordinated sponsorship of both NDE system and software development have nurtured numerous technological advancements. In fact, the AEP-sponsored effort to replace ANSYS by nonproprietary thermal and stress analysis codes forms the nucleus of improvements in the forthcoming second generation of STRAP.

The process of a run/retire decision involves engineering design and analysis and fracture mechanics disciplines in addition to the key role played by the non-destructive inspector. The details in Parts 2 and 3 of this STRAP software manual will be of interest to these people and, because the decision to retire a steam turbine rotor necessarily involves utility upper management, the less technical overview in Part 1 can help provide them with the required background for a sound decision.

Floyd E. Gelhaus, Program Manager
Nuclear Power Division

ABSTRACT

The automated Steam Turbine Rotor Analysis Programs (STRAP) have been developed to facilitate the prediction of rotor lifetime given the duty cycle of the turbine and the results of ultrasonic examination from the rotor bore. STRAP consists in part of a preprocessor that generates the boundary conditions and finite element mesh for transient and steady-state temperature and stress analysis. The input thus generated is utilized by ANSYS, a general purpose, finite element structural analysis program. A postprocessor within the STRAP system contains fracture toughness, stress-rupture, yield strength, and fatigue crack growth rate data for air-melted CrMoV forgings, on the basis of which ANSYS-calculated stress and temperature values are screened to determine critical crack size, initial crack size that could grow to critical size within a specified number of hours or cycles, and minimum area fraction of defects which could link to result in a significant crack. A boresonic data reduction program allows rapid scanning of indicated flaw sizes and locations to find the regions of greatest defect density.

ACKNOWLEDGMENTS

The authors would like to acknowledge Janet Mann and Debbie Jameson from Southwest Research Institute, along with Mark Oakely and Tom Gruber at Battelle Columbus Laboratories for their contributions in programming STRAP updates and aiding in the documentation of this analysis system.

TABLE OF CONTENTS

LIST OF ILLUSTRATIONS

SUMMARY

A Steam Turbine Rotor Analysis Program (STRAP) has been developed to allow the user to evaluate the remaining service life of a rotor given the results of ultrasonic examination from the bore. The computer programs within STRAP consist of a preprocessor (PPMESH), a postprocessor (FRAC), and an ultrasonic data processing subprogram (CLUELP). The ANSYS finite element program is employed as an intermediate link between the PPMESH and FRAC programs to calculate transient temperature and stress distributions. ANSYS is external to the STRAP system, and the source level encoding is not available for modification.

The preprocessor, PPMESH, has been specifically written to generate a finite element structural model of either a General Electric or Westinghouse rotor. Given the rotor dimensions and the operating history of steam temperature, steam pressure, and rotor speed, the PPMESH program generates sufficient data for the calculation of transient and steady-state temperature distributions throughout the rotor. The actual results are obtained by processing the PPMESH data via the ANSYS program. The temperature solution thus obtained is further processed by ANSYS to calculate the stresses at the centroidal of each finite element of the rotor model.

FRAC uses as input ANSYS-generated transient and steady-state stress values along with either the flaw dimensions calculated by CLUELP from the nondestructive examination results, or with arbitrary, user-specified flaw sizes and locations. FRAC calculates the length to which a fatigue crack will grow from a user-specified initial length within a specified number of cycles. The CrMoV material properties necessary for calculations have been incorporated into the FRAC program. At the user's option, FRAC will also calculate both the minimum critical crack size and the required initial crack size which would grow to a critical size by fatigue within the specified number of cycles. These calculations can be performed for up to ten discrete flaw locations in a single computer run; four types of flaw geometries are available for such computations.

CLUSTR, a subroutine called by CLUELP, analytically predicts the linking between neighboring ultrasonic indications, given their measured sizes or equivalent flat-bottomed hole diameters based upon amplitudes. The linking analysis considers the interaction distance and angle and the possibility of ligament plastic instability or stress rupture. FRAC utilizes these linked-up flaw configurations as initial cracks for a conservative calculation of remaining service life of the rotor.

1.0 INTRODUCTION

Electric power utilities need the capability of predicting remaining service life of steam turbine rotors. These rotors generally contain flaws resulting from either manufacturing or operational history and are detected by inservice ultrasonic inspection from the bore surface. An accurate prediction of remaining service life of the rotor requires the application of several areas of engineering discipline. First, there must be nondestructive examination (NDE) data of sufficient quality to define the location and size of flaws. Next, a transient temperature and stress analysis of the rotor must be performed, simulating as closely as possible the anticipated operating condition of the rotor. In order to perform these analyses, a data base containing physical properties of the rotor material under operating conditions is necessary. Once the NDE data and the computed stress and temperatures are available, the remaining service life can be estimated from fracture mechanics calculations of crack growth.

This manual offers a general overview of the computerized lifetime prediction analysis system called STRAP. This system consists of three computer programs, a preprocessor (PPMESH), a postprocessor (FRAC), and an ultrasonic inspection data analysis subroutine (CLUELP). A commercially available, general-purpose finite element analysis code, ANSYS*[(1)], is used as an intermediate link between the PPMESH and FRAC programs. Details of these various programs are described in Part II, User's Manual and Part III, Programmer's Manual.

---

*Note: Number in parenthesis corresponds to an item in the list of references in Section 4.0.

2.0 ANALYSIS METHODS

A brief description of the methods for performing heat transfer analysis, thermomechanical stress analysis, and fracture mechanics analysis that are contained in the STRAP system is as follows.

2.1 Heat Transfer Analysis

Transient and steady-state temperature distributions within a steam turbine rotor are determined via the ANSYS finite element program. Typical input data required for performing this analysis are:

- Finite element mesh of the rotor geometry;
- Convective heat transfer coefficients for convective surfaces;
- Equivalent thermal resistance for heat conduction from the blading to the disk rims;
- Thermal analysis time steps for a discretized solution in the time domain; and
- Thermophysical properties of CrMoV steel and of steam.

All the above data are provided by the PPMESH preprocessor. The heat transfer coefficients depend upon steam leakage flows and seal configurations, rotor surface speed, and steam properties. The expressions for convective heat transfer coefficients are given in Part III of the STRAP manual. In addition, an equivalent resistance for heat conduction from the blading to the disk rims is also specified by the PPMESH program. The transient thermal analysis thus requires the user to specify seal geometry and steam inlet and outlet temperature and pressure as functions of time. The physical properties of steam are built into PPMESH.

A single seal geometry giving the maximum leakage flow (minimum number of teeth, maximum clearance) may be specified since the transient rotor temperature is relatively insensitive to seal geometry. Leakage flows are calculated from Martin's equations(2) and the heat transfer coefficients on the rotor surface from the correlations given by Kapinos and Gura.(3) Other rotor sections are modeled as rotating cylinders and disks. It should be noted that while the description of the boundary conditions via PPMESH is considered reasonably complete, it does not contain certain important factors, such as secondary flows and degree of turbulence, which may differ from those developed in the experiments upon which the correlations used for disks and cylinders were based.

The duty-cycle representation usually consists of three parts: prewarming, roll-off to synchronization, and loading from no-load to full-load. PPMESH does not contain the steam expansion lines from the Mollier diagram, but distributes the pressure and temperature drops across each reaction stage (according to experimental extraction point data) given the time variation of inlet and outlet steam conditions. The experimental data were obtained from General Electric and Westinghouse turbines, and are adjusted for different percentage drops across the stator and rotor blading. STRAP (PPMESH) will accept any arbitrary history of steam conditions and rotor speed as input. The resulting temperature distributions are stored on any input-output computer device (e.g., magnetic tapes, disk files, punched cards) for further processing. In this case, the objective is to calculate the corresponding stresses (thermomechanical stress analysis).

2.2 Thermomechanical Stress Analysis

STRAP utilizes the ANSYS program to determine the distribution of centrifugal and thermal stresses throughout the rotor volume, with PPMESH providing the necessary data. The stress analysis requires estimates of blade weight for each stage, which are supplied by the user. Centroidal values of tangential stresses are calculated for all the elements in the PPMESH-generated mesh and stored with the corresponding temperature at each time step. STRAP contains interpolation and extrapolation algorithms to determine the stress at any radial and axial location within the rotor. The stress analysis does not include inelastic material response. Since the mesh generation was designed to provide accurate stress distributions only within a 4-inch radial distance from the bore surface, the stresses near the periphery of the rotor are not sufficiently accurate for calculations of diaphragm groove or disk rim cracking.

2.3 Fracture Analysis

The FRAC code of the STRAP system provides a number of computations for possible failure modes: brittle fracture, ductile fracture, low-cycle fatigue crack growth, and stress rupture. FRAC does not contain provisions for stress corrosion cracking, creep crack growth, or for cracking in bending or torsional fatigue; however, methods are contained for predicting the growth and criticality of planar flaws of various shapes. The current procedure for determining rotor integrity from boresonic results is to reduce each indication to an area on a radial-axial plane and to assume the regions of high area fraction denote flaws with the dimensions of an ellipse circumscribing that region. Such a procedure is conservative and very subjective. This conservatism, along with the lack of confidence in amplitude data, prompted the inclusion in STRAP of an indication scaling factor (which is a function of local tangential stress and yield stress) to determine the interaction distance between distributed flaw indications. All flaw indications which fall within an interaction search length are treated as a single flaw with dimensions of an ellipse circumscribing the flawed region.

While FRAC does not incorporate nonlinear material behavior, it does contain a conservative criterion for cluster linkup taken from the work of Melville.(4) He states that when the average ligament stress equals the yield stress, the ligaments are assumed to fail, leading to an "unzipping" process at overspeed during a single cycle. This criterion is very conservative since it is known that such clusters can resist many cycles of reversed plastic strain before linkup occurs. The user can compare the calculated area fractions and initial crack sizes by FRAC with boresonic data to evaluate the severity of the cluster.

2.4 Critical Crack Size Calculation

The critical value of stress intensity factor, K_{IC}, or of path-independent J-integral, appears to correlate with the unstable propagation of a crack and has been built into FRAC as the first step in fracture analysis. Because of the possibility of overspeed startup and in view of the requirement to test overspeed trips, a nominal overspeed of 15 percent of synchronous is assumed in FRAC; however, any other value can be specified by the STRAP user. The values of critical crack size are calculated for flaw coordinates, and a variety of flaw aspect ratios. The flaw coordinates and aspect ratios are determined by a CLUELP analysis of the boresonic data. If inspection data are not available, FRAC determines the axial location of the minimum critical flaw size and its variation with distance from the bore surface, thereby specifying the areas over which the boresonic data reduction must be carried out.

The location of the minimum critical crack size usually coincides with that of the maximum tangential stress; however, it is possible that lower-stressed regions might exhibit a smaller value because of lower fracture toughness. Lower toughness may result from the local stress peaking at a lower temperature or from temper embrittlement of those portions of the rotor subjected to high service temperature. The FRAC Code examines such areas to ascertain the minimum critical crack size.

3.0 DESCRIPTION OF THE STRAP SYSTEM

This section describes the various programs that are part of the STRAP System.

3.1 PPMESH Program Description

The PPMESH program prepares and manages the input necessary to perform a thermal and/or stress analysis using the ANSYS program. Figure 1 shows the logic flow between these two programs. Since PPMESH serves as a primary input executive for the ANSYS program, several tasks are performed by it. The following sections contain the data that PPMESH requires from the user for a successful program execution. Also, the procedure for input data reduction is discussed.

3.1.1 Functions and Options of the PPMESH Program

The following functions are performed by PPMESH:

- Defines all variables and rewinds all disk files;
- Reads all input needed by the preprocessor;
- Generates finite element nodal coordinates;
- Generates element definition for the finite element map;
- Calculates the heat transfer coefficients for the convection surfaces of the rotor;
- Calculates forces exerted by the blades on the rotor; and,
- Writes files needed as input to the ANSYS analysis run.

The following are the user-controlled options. The user may:

- Generate the mesh for a Westinghouse rotor;
- Generate the mesh for a General Electric rotor;

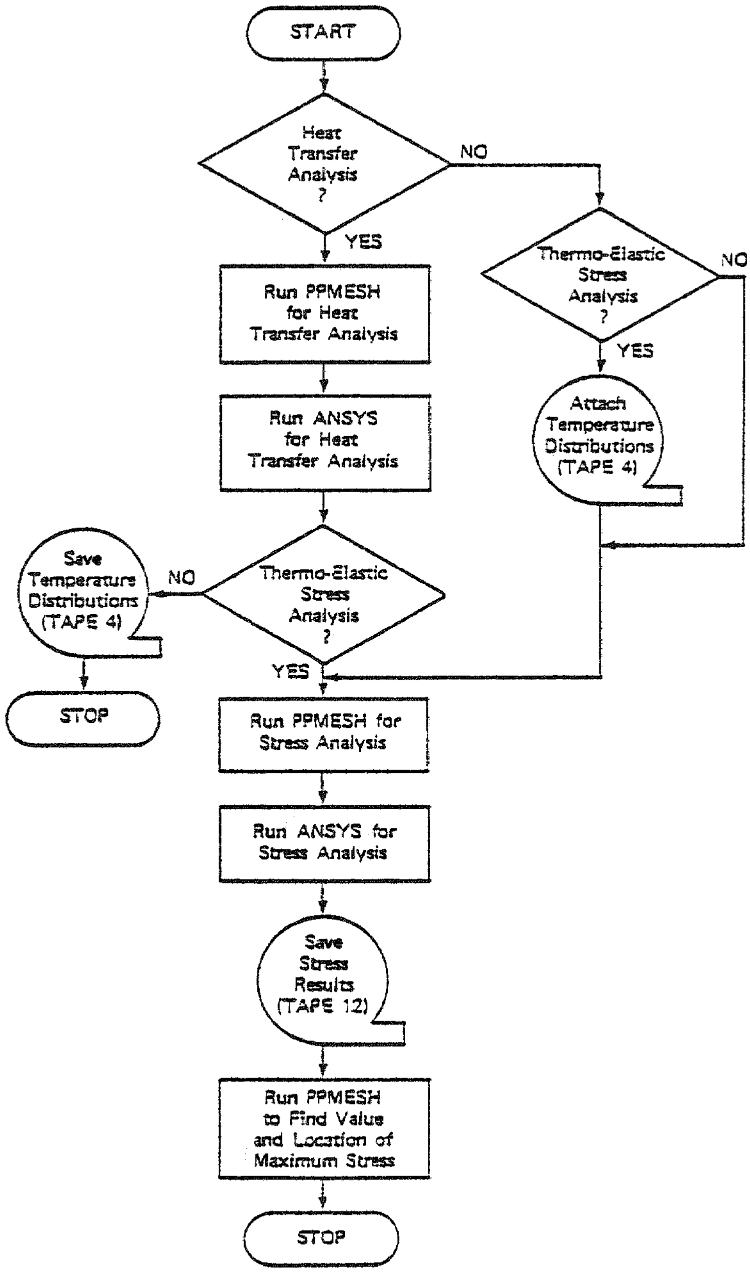


FIGURE 1. INTERCONNECTION OF PPMESH WITH ANSYS

- Read blade masses, lengths, radii, and blade spacings from disk file if back-to-back runs are being made;
- Perform a heat transfer analysis;
- Perform a stress analysis;
- Use temperature distributions from a previous heat transfer analysis as input to a thermomechanical stress analysis;
- Change heat transfer coefficients on rotating shafts to heat transfer coefficients for labyrinth seals;
- Choose to supply temperature boundary conditions at either of the extreme axial ends of the rotor - ordinarily, these boundaries are considered insulated; or
- Interrogate the stress files and determine the magnitude of the maximum tangential stress, the location, and the load step on which it occurred.

3.1.2 Mesh Generation Techniques

The basic finite element maps of the General Electric and Westinghouse rotors are shown in Figures 2 and 3. These figures show coordinate systems, examples of critical nodes, and general features of the idealization of the two rotor types.

The limitations (reference Figures 2 and 3) of the mesh generation technique are:

- $RR(1) > R(1)$;
- Rotor must have a constant bore diameter,
- $RR(3) = RR(4)$; and
- $R(1) \leq R(2) \leq R(3) \leq R(4) \cdots \leq R_{NSTAGE}$.

UZ(Axial Displacement) = Fixed Nodes 1 through 8
RC = Distance from X = 0 to Node 381
R(1) = Distance from X = 0 to Node 71
RS(1) = Distance from X = 0 to Node 69
R(NSTAGE) - Distance from X = 0 to Node 345
RS(NSTAGE) = Distance from X = 0 to Node 342
DB = Bore Diameter
HC = Heat Transfer Coefficients

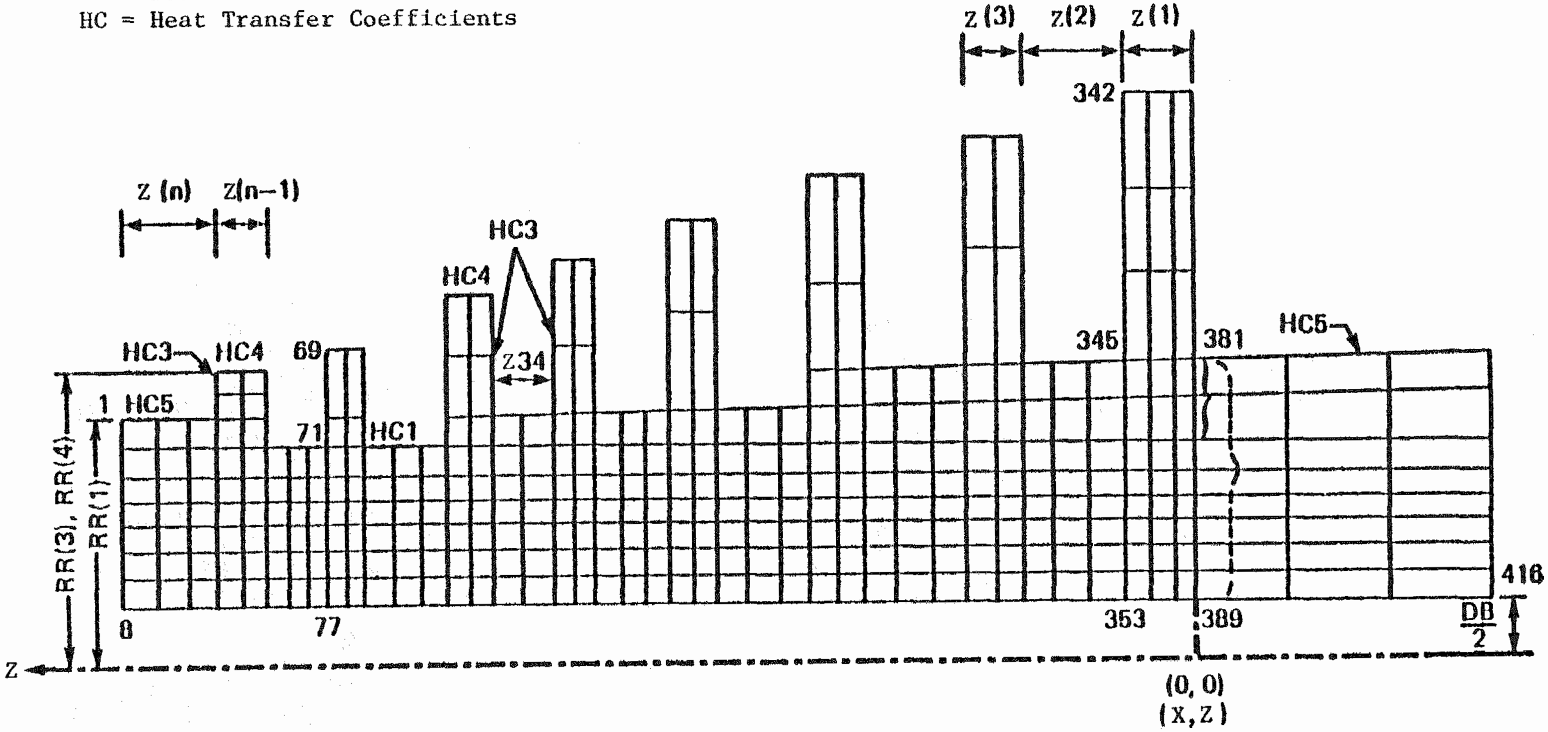


FIGURE 2. SCHEMATIC OF ROTOR MESH: GENERAL ELECTRIC ROTOR

UZ(Axial Displacement) = Fixed for Nodes 1 through 8
RC = Distance from X = 0 to Node 443
R(1) = Distance from X = 0 to Node 57
RS(1) = Distance from X = 0 to Node 55
R(NSTAGE) = Distance from X = 0 to Node 386
RS(NSTAGE) = Distance from X = 0 to Node 383
RF = Distance from X = 0 to Node 439
RSH = Distance from X = 0 to Node 448
DB = Bore Diameter
HC = Heat Transfer Coefficients

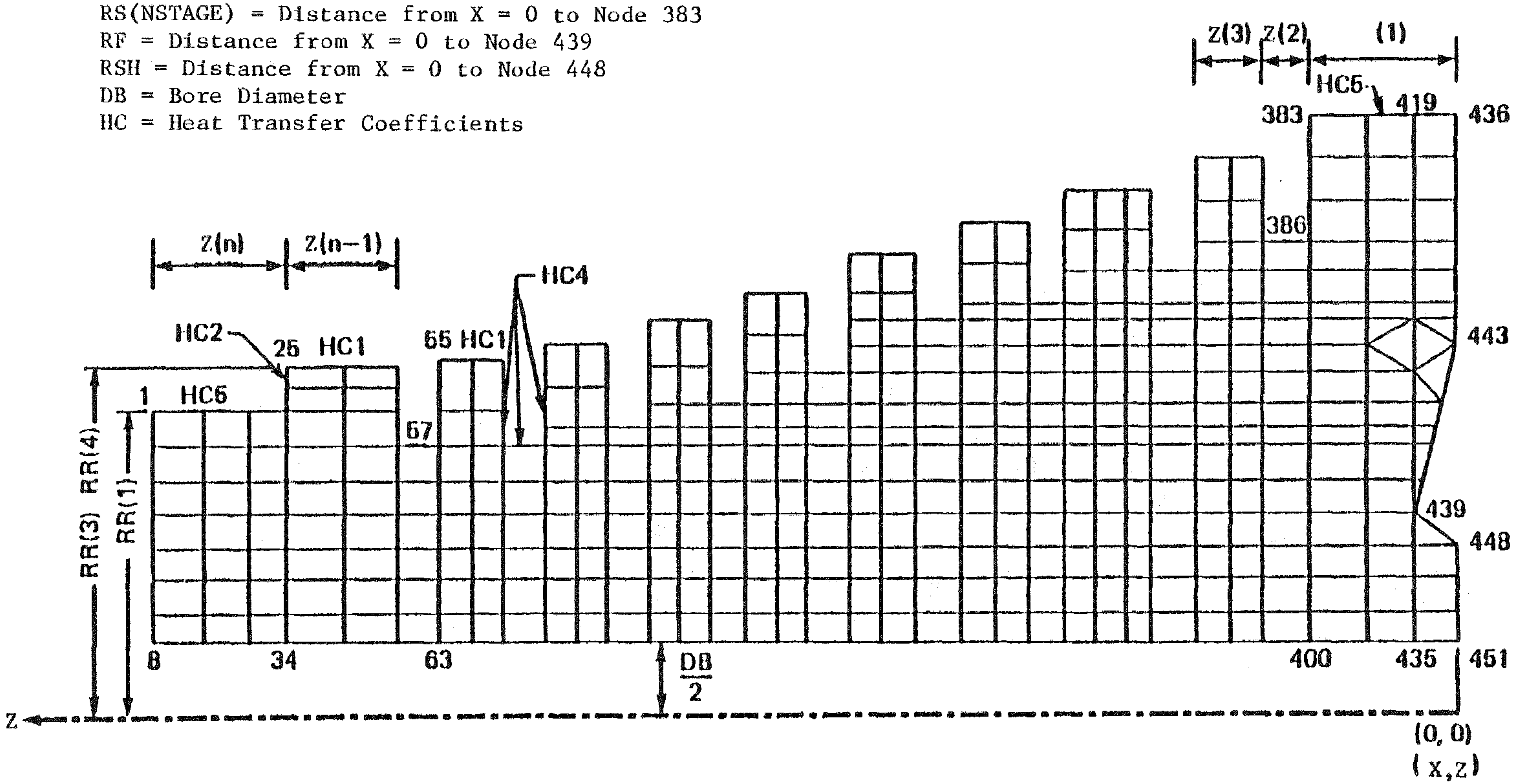


FIGURE 3. SCHEMATIC OF ROTOR MESH: WESTINGHOUSE ROTOR

3.1.2.1 Choice of the Element Type and Size

A higher order, four-node, two-dimensional isoparametric element is chosen from the ANSYS program finite element library. However, the user has no direct control on the shape functions (displacement polynomials) that are built into ANSYS finite element stiffness subroutines. The shape function used to determine this particular finite element is as follows:

$$N_i = a_1 + a_2x + a_3z + a_4zy, \qquad (1)$$

where

a_i = nodal constants ($i = 1,2,3,4$);

x,z = nodal coordinates; and

N_i = shape function.

The displacement (U,V) field at each nodal location is given by the following equations:

$$U = N_1U_1 + N_2U_2 + N_3U_3 + N_4U_4, \text{ and} \qquad (2)$$

$$V = N_1V_1 + N_2V_2 + N_3V_3 + N_4V_4. \qquad (3)$$

The consequence of using such an element is that the strain is not restricted to a constant; therefore, fewer elements and less computation effort result in a converged solution when compared to the constant-strain finite elements.

3.1.2.2 General Electric Rotor Geometry

The basis for the element generation of the General Electric rotor is the radial distance between nodes 71 and 77 in Figure 2. This distance is divided into six equal divisions when the radius of node 381 is greater than or equal to the radius of node 71. When the radius of node 381 is less than the radius of node 71, the radial distance between the bore (node 77) and node 381 is divided into three equal divisions.

In the finite element idealization of the General Electric rotor, the interstage blade spacing is modeled one element wide unless this dimension is greater than or equal to 2.5 inches. When this occurs, two elements are used for spacing up to 3.21 inches and three elements for larger distances. Three elements are shown in Figure 2 for each spacing except for one, indicated on the figure by Z34. In a similar manner, the General Electric blade root widths are divided into two elements; when the spacing is greater than 3.21 inches, the spacing is divided into three elements. Two elements are shown in Figure 2, except for the attachment length denoted Z(1).

The axial dimension of the element depends on the blade spacing. However, the two regions at the ends of the rotor require special attention. At the inlet end of the rotor, the length Z(n) of the cylindrical section is divided into three equal increments. While the user is free to specify this length in modeling the section, Z(n) should be at least equal to the radial distance between nodes 71 and 77. At the outlet end, the program generates a cylindrical section which is divided into three axial increments, each being equal in length to the radial distance between nodes 345 and 353. This cylindrical section is used to simulate the thermal boundary conditions experienced at the matching boundary between combined IP and LP rotors when the IP rotor is the part of the rotor of interest and the other rotor half, i.e., the LP, is not to be modeled. When just the LP rotor is modeled, this cylindrical section is used to simulate the thermal boundary conditions experienced at the matching boundaries between the LP rotor and the trailing cylindrical shaft. No detailed stresses are given in this cylindrical section.

3.1.2.3 Westinghouse Rotor Geometry

For the Westinghouse rotor, the radial distance between nodes 57 and 63 is divided into six divisions, as shown in Figure 3. These divisions are determined as follows: the radial thickness of the shaft first is equally divided into three elements, and then the radial distance between the middle of the arc in the fillet region and the bottom of the first stage is divided in half, giving two more radial increments. These dimensions, coupled with the coordinates of nodes 448 and 439, fix the radial coordinates of the first six radial divisions of the structure, thus giving three critical input parameters which govern these six divisions. These parameters are:

- The radius to the bottom of the first blade attachment - node 57;
- The radius to the middle of the arc in the fillet region - node 439; and
- The outer radius of the shaft - node 448.

The axial dimension for the cylindrical section at the inlet end of the rotor Z(n) is divided into three equal increments. In modeling this section, the axial distance between nodes 8 and 34 in Figure 3 is made at least equal to the radial distance between 57 and 63.

In the idealization of the Westinghouse rotor, the interstage blade spacing is two elements wide; however, if the spacing is greater than 3.21 inches, the spacing is divided into three elements. Similarly, the blade attachment is considered to be one element in length unless this dimension is greater than or equal to 2.5 inches. Two elements are used for root widths between 2.5 and 3.21 inches, while three elements are needed for widths exceeding 3.21 inches. The attachment region above the next blade cut-out is always divided radially into three finite element lengths.

3.1.3 Duty-Cycle Input Data

For the heat transfer analysis, all the required physical properties of steam are built into the preprocessor. However, there are five types of user-specified input data needed to describe the duty cycle. The user must:

- Specify the RPM-time history;
- Specify the fraction-of-load as a function of time;
- Specify the sealing steam temperature time history;
- Input the inlet and outlet pressure for three set times during the duty cycle; and
- Specify the inlet and outlet steam temperatures.

3.1.4 Pressure Data

Three pressure conditions are input to describe the pressure-time history, as shown in Figure 4: the inlet and outlet pressures during roll-off (the inlet pressures before roll-off are assumed equal to the inlet pressures during roll-off), at synchronous speed with no load, and at maximum load. Prior to and during roll-off, the pressure drop along the rotor is assumed to be zero. However, because a zero pressure drop causes computational problems, the inlet and outlet pressures must differ by some small quantity, which should be equal to 0.25 psia. Following roll-off, as the rotor is brought up to operating speed, the pressure drop is assumed constant; also, the inlet and outlet pressures remain constant. After synchronous speed is attained, a new set of pressures is applied. The final set of input pressures, the inlet and outlet pressures at maximum load, is scaled to give the pressure for any partial load condition after rotor synchronization.

A linear relationship exists between pressure drop and load; thus, if the pressures at maximum load are given, the pressure history for any subsequent load history can be obtained. Once the rotor is at synchronous speed, the pressure drop along the rotor is calculated; this pressure drop is based only on the inlet and outlet pressures. Using the limited amount of pressure measurements obtained from inservice rotors, it was determined that a linear distribution was the best approximation to the pressure drop along the Westinghouse rotor. Similarly, a limited data set received for the General Electric Joppa #3 rotor indicated a negative exponential distribution would best describe the pressure drop along that rotor. Later experience indicated that the linear pressure drop was conservative and adequate for subsequent analysis. Therefore, the linear pressure drop has been incorporated into PPMESH for all analyses.

3.1.5 Temperature Data

The temperature fields are specified at six times (or in the case of the sealing steam, seven times) throughout the operating cycle. The temperature at intermediate times is obtained by interpolation of these values. The inlet and outlet steam temperatures define the temperature drop along the rotor. On the basis of limited operating data, this temperature drop is assumed to be linear along the General Electric rotor while a sine-square function is used to describe the drop along the Westinghouse rotor. The sealing steam temperature

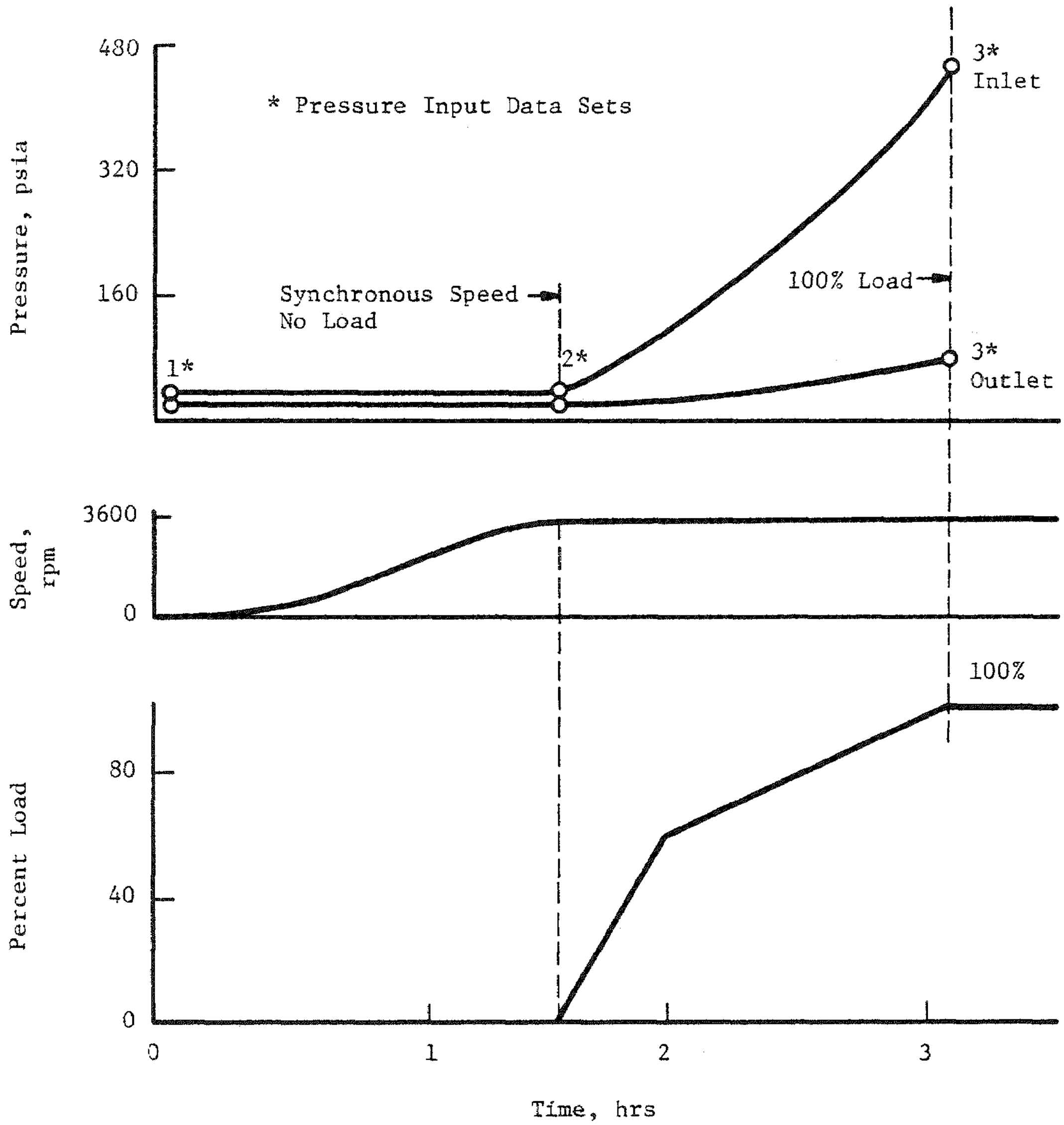


FIGURE 4. PRESSURE INPUT DATA (SCHEMATIC)

is used to determine the temperature of the steam or water seals, and the RPM-time history is used in the calculation of the heat transfer coefficients for shaft elements that are treated as rotating disks.

3.1.6 Interstage Seal Heat Transfer

While it is not the intent of this program to analyze the periphery of the rotor in detail, it is necessary, nevertheless, to consider the interstage seals in some detail in order to compute the pressure drop and leakage flow through these seals for an adequate calculation of the heat transfer coefficient. Since detailed seal geometry information is not generally available, it was necessary to assume that the pressure drop across the seals is some fraction of the total interstage pressure drop. It was assumed that for the Westinghouse and General Electric (GE) rotors, 50 and 75 percent, respectively, of the pressure drop for each stage took place across the labyrinth seals. This pressure drop is used in computing the seal leakage flow by Martin's formula(2).

It should be noted that there is a difference here between the GE and the Westinghouse rotors. The seal geometry, i.e., the number of teeth, in the Westinghouse rotor is built into the program, as it is the same at different locations of the rotor. For the GE rotor, the seal geometry is different at different rotor locations. Thus, the number of seal teeth must be specified by the user. Experience with the program has shown, however, that the temperature distribution is only slightly perturbed by changing the seal geometry.

3.1.7 Thermal Analysis Time Steps

During the transient heat transfer analysis, two procedures are used for choosing time steps. The first procedure is always used for the first load step and any other load steps related to the rotor warmup. This procedure is simply an ANSYS-contained convergence scheme which allows the size of the time steps to increase from as small as six minutes to as large as possible as the solution for that particular load step is reached. (Load step is defined to mean any time a complete set of boundary conditions is specified so that a solution may be obtained.) The second procedure includes a refined time period, which is simply a fixed time step of six minutes, that is used to refine the time during and after the loading cycle in order to determine the maximum tangential stress within three minutes of occurrence. See Figure 5 for an explanation of the time step refinement. This choice of time increments resulted from prior experience with stress analysis of the Gallatin #2 IP and the Joppa #3 IP-LP rotors. The maximum tangential stress in both rotors was found to occur in the first 1.5 hours of load time. Since this program is intended to determine the maximum rotor stresses, a two-hour time period was chosen for refinement and was considered sufficient to include these peak stresses. However, it should be emphasized that this two-hour period is fixed as being the last two hours of the particular cycle under study. Although the user does not have the option to vary the position of the refined period, he does have the option to vary the length of this refined period in case a refined stress history of some other portion of the operating cycle is required. Of course, the maximum stresses can be determined during any period, but it is only within the refined time period that the stresses are computed frequently enough to alleviate the possibility of overlooking a local stress maximum.

3.1.8 Stress Analysis Boundary Conditions

When a stress analysis is desired, the preprocessor (PPMESH) calculates and applies all necessary mechanical boundary conditions. The rotor is fixed at the inlet end against axial displacement. The blade forces are calculated using the input blade masses and multiplying them by the appropriate radial acceleration. This force is divided by 2π in order to express it in pounds force per radian, which is necessary in an axisymmetric analysis. Half of this force is applied to each side of the blade attachment area.

3.2 FRAC Program Description

The FRAC computer program has been written to assess, through fracture mechanics analysis, the remaining life of a steam turbine rotor containing specified flaws or inspection indications. It performs this task by combining the results of the thermomechanical stress analysis of the rotor, typical materials data, and the results of nondestructive examination (NDE) to pinpoint critical rotor locations and to evaluate quantitatively the severity of designated flaws. The designated flaw indications are calculated from the NDE data by CLUELP. The FRAC is generally based on linear elastic fracture mechanics, but it does incorporate other concepts (e.g., the Larson-Miller parameter is

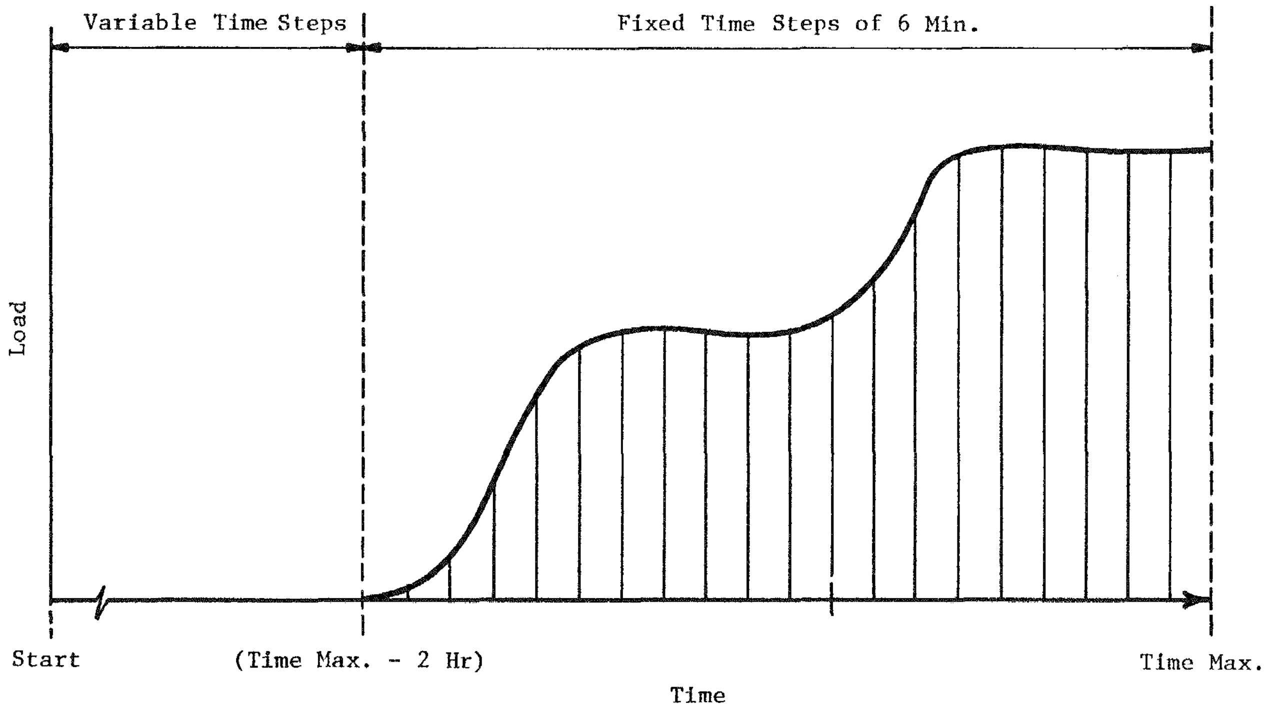


FIGURE 5. TIME STEP REFINEMENT

used to evaluate the creep-rupture life). Figure 6 shows the flow of the structural analysis and indicates the position of FRAC in the total analysis scheme.

Detailed flow charts for FRAC and its subprograms are given in Part III, Programmer's Guide,[5] and its use is described in Part II, User's Manual.[6] Briefly, the essential steps of the code are as follows:

(1) Based on the thermomechanical stress analysis, the smallest crack that can result in unstable fracture during an overspeed is determined;

(2) If, as a result of NDE, specific locations in the rotor are of interest, the overspeed crack size that will cause fast fracture at these locations is computed;

(3) The initial crack size that can grow to this critical crack length by fatigue in a designated number of operating cycles is determined;

(4) The potential for creep-rupture and linkup between defects by ligament yielding is examined through a simple area fraction model; and

(5) The remaining life is predicted by calculating the number of cycles to failure for each designated flaw.

This program either provides the basis for judging the potential of crack growth from specific NDE indications or determining what areas of the rotor are most critically stressed and, therefore, should be examined most carefully. Flaw growth is evaluated on the basis of anticipated usage, i.e., the specific duty cycle and the number of cycles the rotor will be subjected to during its lifetime. While the program has been formulated in a conservative manner, the quality of the input data plays a dominant role in the accuracy of the results. Thus, the user is cautioned to examine the input duty cycle data used in PPMESH carefully and to make every effort to ensure the input data actually represents the operational duty cycle.

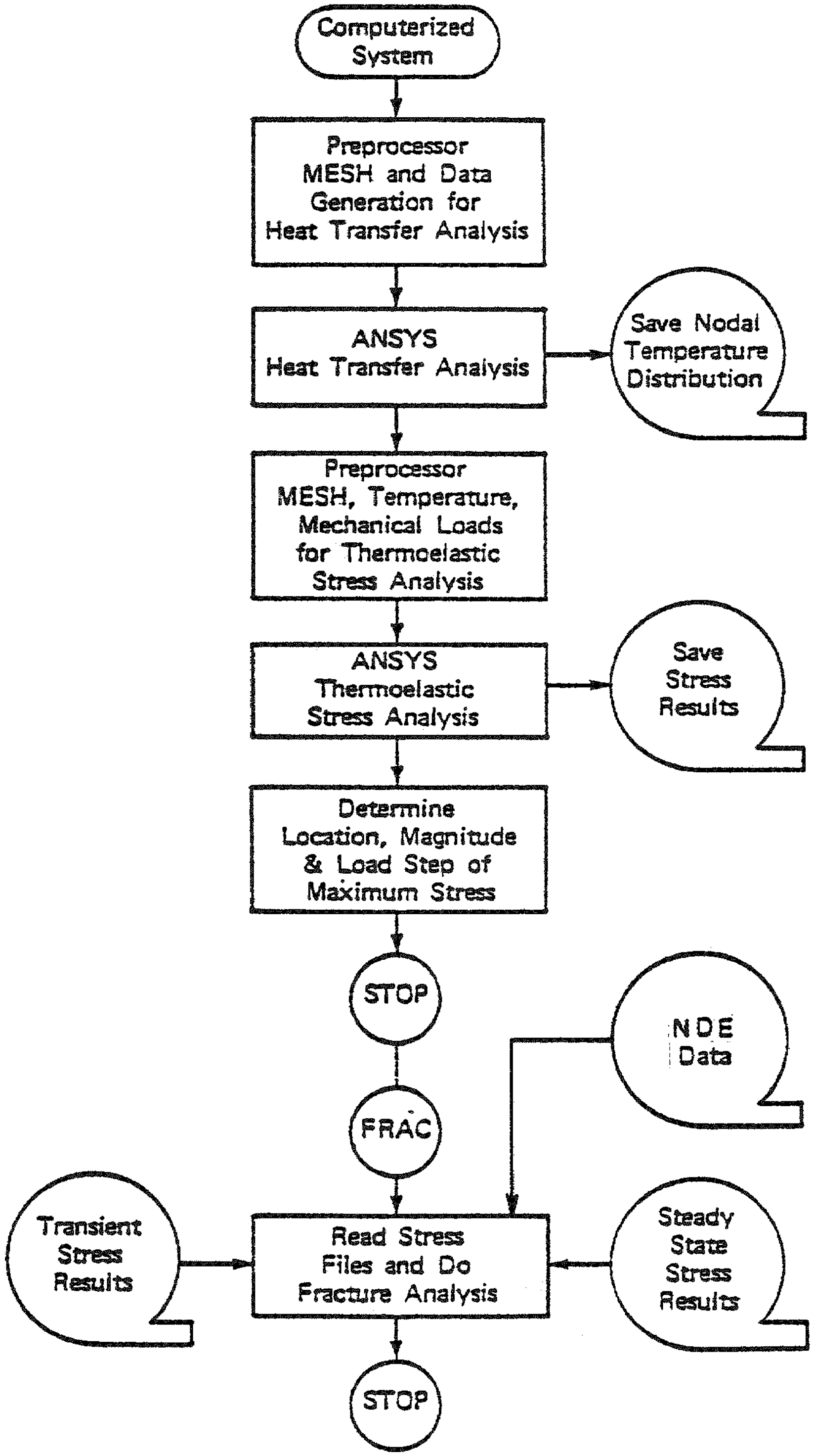


FIGURE 6. FLOW OF COMPUTERIZED SYSTEM

3.2.1 Critical Crack Size Calculation

FRAC analyzes the subsurface flaw as a circular or elliptical crack, while the surface flaw is analyzed as a long bore-connected crack with a small aspect ratio. The program can handle up to ten different discrete flaw locations in addition to the single critical location. In general, the critical crack radius is given by

$$a_c = C \left(\frac{K_{IC}}{\sigma}\right)^2 \qquad (4)$$

where

a_c = critical crack radius, (inches);

C = a geometric constant determined by the crack geometry;

K_{IC} = the material fracture toughness at the given temperature, (ksi $\sqrt{in}$); and

σ = the tangential stress at the point of interest, (ksi).

The range of material fracture toughness as a function of temperature is defined by the rotor's operating history and the manufacturer's forging procedure. After the toughness data is gathered, the local metal temperature in the area of interest is used to determine K_{IC}.

3.2.2 Initial Crack Size Calculation

Having determined the critical crack size, FRAC next calculates the initial crack size that could grow to critical size with a user-specified number of duty cycles. Also, CLUELP specifies an initial crack size, based upon the boresonic results, and FRAC calculates the number of cycles required to reach critical size. The size thus calculated represents an isolated, crack-like flaw in a radial-axial plane that could grow to critical size in the calculated number of duty cycles. The Paris law is used in these crack growth calculations.

3.2.3 Cluster Analysis

A general and complex problem in the steam turbine rotor lifetime analysis is the assessment of three-dimensional arrays of individual defects commonly called clusters. Predicting the linkup between distributed flaws in a cluster requires several steps:

(1) Determination of the interaction distance between neighboring flaws, especially that distance in the circumferential direction normal to the radial-axial plane of fracture over which flaws link up;

(2) Calculation of progressive linkup by fracture mechanics where flaw sizes are known from boresonic data; and

(3) Determination of the criteria for inelastic instability by ligament yielding and for ligament/creep rupture.

3.2.4 Fracture Analysis Procedure

Thus, the initial program operation is to determine the extreme values of the temperature and the stress. By making an efficient search of the transient stress analysis data files, the location of the maximum rotor stress is determined. At the same time, the program stores the radial distribution of temperature and stress at each time step for each discrete flaw location. Once the temperature and stress data have been accumulated at the desired locations, the program is in a position to determine the critical crack sizes at these locations.

It has been pointed out that a critical crack size is computed at the location of maximum stress. While this location frequently yields the minimum crack size, the decrease in fracture toughness with temperature below FATT means that minimum critical crack could occur at lower stress in a low temperature section of the rotor. Therefore, the cooler end of the rotor, which is considered to be the last ten to twelve inches at the exhaust end, is also searched for the maximum stress. After processing all time steps, minimum crack size in the entire rotor is determined by comparing the results from exhaust end calculations and maximum rotor stress calculations. After the location and value of the minimum critical crack size for the entire rotor is determined, stress and temperature distributions as a function of radial distance for this location are saved.

The program can compile the data for sixteen time steps at ten different discrete flaw locations.

Following processing of transient data, steady-state data are read and stored for all locations of interest. With the steady-state data and transient data resident in the computer memory, the critical half-crack length for each location of interest is recalculated based upon a percentage of overspeed, which is input by the operator. The overspeed stress is calculated by the expression

$$S = TS + SS\,(OS^2-1), \tag{5}$$

where

S = the overspeed stress, (ksi);
OS = ratio of overspeed rpm to synchronous rpm;
SS = the steady-state stress, (ksi); and
TS = the transient thermomechanical stress at synchronous speed, (ksi).

Then the critical half-crack length is calculated.

For the discrete flaws, FRAC calculates: (1) critical flaw size, based on overspeed; and (2) initial half-flaw length needed to grow to that critical size by fatigue under normal synchronous speed operation. Normal operation is defined as a specified duty cycle without any overspeed occurrence during the cycle. FRAC also calculates a final crack radius that can grow by fatigue from an initial crack size determined by a CLUELP analysis of the boresonic data. Steady-state temperatures at flaw locations of interest are used in all fatigue calculations; since steady-state temperatures are higher, this results in the fastest crack growth. However, stresses used to grow the postulated crack are conservatively assumed to be maximum transient stresses at each location.

FRAC also calculates critical half-crack length as a function of radial distance for the axial location of the minimum critical crack. Once this is done, the minimum critical crack radius, the initial crack radius that can grow to critical, and the final half-crack length grown from a specified initial length are calculated.

Failure, by stress rupture or creep-fatigue interaction, of the ligaments joining a planar cluster of inclusions is predicted through the use of a modification of the Larson-Miller parameter(7). This analysis results in the inclusion area fraction

$$\frac{A_i}{A_c} = \left(1 - \frac{\sigma}{\sigma_{LM}}\right), \tag{6}$$

where

A_L = the sum of the cross-sectional areas of all inclusions contained in the planar cluster, (inches^2);

A_c = the area enclosed by the geometry circumscribing the planar cluster of inclusions, (inches^2);

σ_θ = local tangential stress, (ksi); and

σ_{LM} = Larson-Miller rupture stress which is given by the data in Figure 7, (ksi).

Figure 7 shows rupture stress versus the Larson-Miller parameter P where

T = local temperature (°F), and

t_R = time to rupture for this material (hours).

3.2.5 Functions and Options of the FRAC Program

FRAC is the main program for fracture analysis modeling. It reads flaw data and critical parameters and reads appropriate stress tables available to the program and calculates and outputs critical crack length data. FRAC estimates the remaining rotor service life.

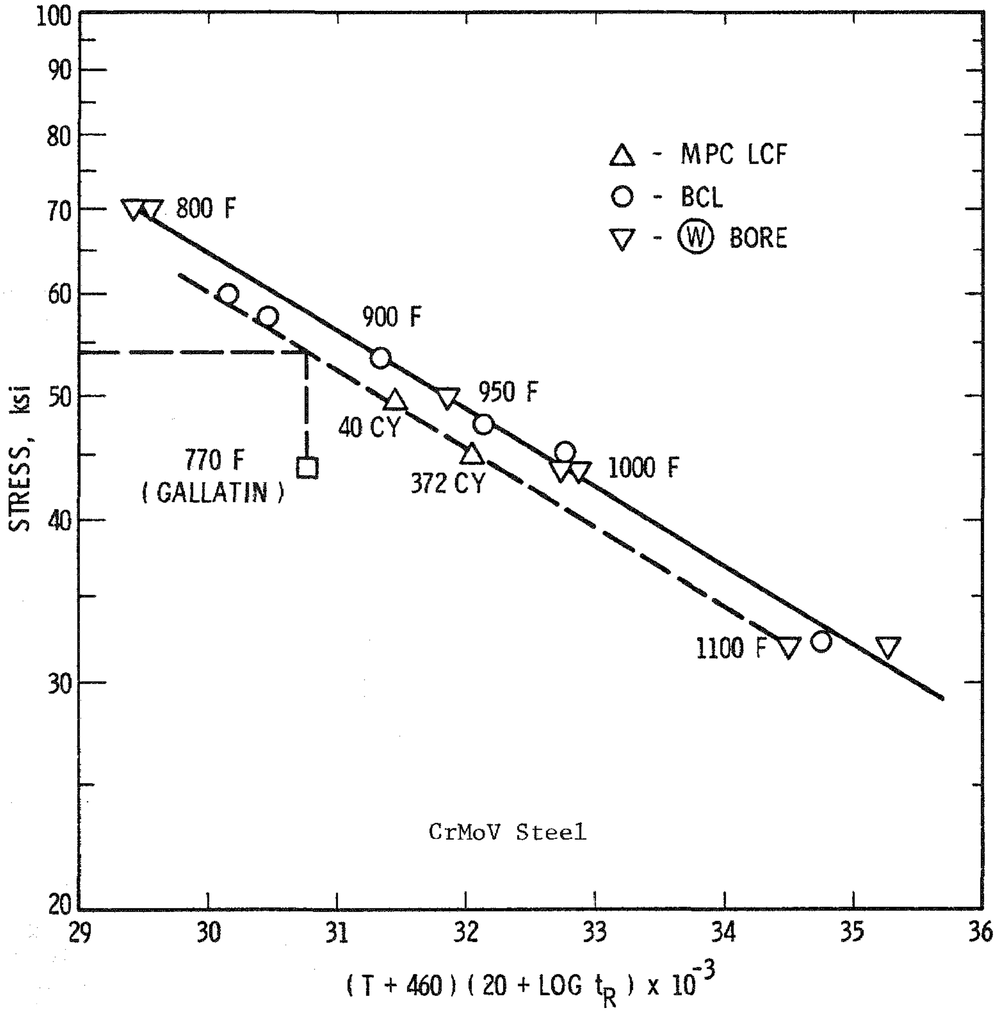


FIGURE 7. LARSON-MILLER REPRESENTATION OF STRESS-RUPTURE DATA

The options available in FRAC are:

- Calculate scaling values used in the linkup analysis;
- Perform a linkup analysis of a three-dimensional array of individual defects;
- Read flaw geometries from cards or from results of linkup analysis;
- Input the number of duty cycles;
- Input value of percentage overspeed;
- Use embrittled or unembrittled material fracture toughness data; and
- Analyze selected sections of the rotor.

3.2.6 Description of CLUSTR and ELIPSE Computer Routines

The purpose of the CLUSTR analysis is to collect ultrasonic flaw indications into three-dimensional clusters on the basis of certain linkage criteria. The ELIPSE analysis defines the cluster in terms of an ellipse enclosing the cluster collapsed onto a radial-axial (r-z) plane for fracture analysis purposes.

3.2.6.1 Description of CLUSTR Routine

The search algorithm of CLUSTR looks forward (higher axial position value) in a half spherical solid angle and computes the vector distance from the current indication to the next indication under consideration. If the vector distance between the two indications is less than the search length, the two indications are linked (Figure 8). If the indication is found in more than one cluster, the clusters are merged to form a single large cluster.

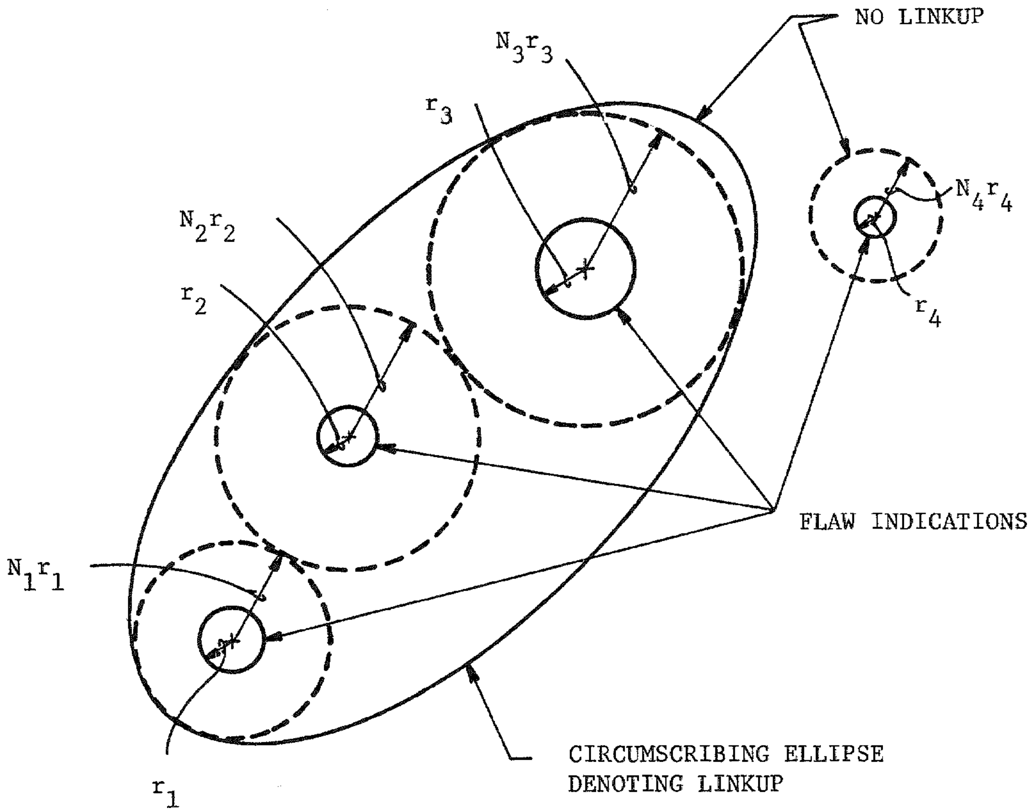


FIGURE 8. SCHEMATIC OF FLAW LINKUP PROCESS

The search length is computed according to the relationship:

$$SL = NR_{cur} + N'R_{cur + m}, \qquad (7)$$

where

R_{cur} is the radius of the current indication, (inches);

$R_{cur + m}$ is the radius of an indication beyond the current indication, (inches); and

N and N' are multiplying factors (N-values) dependent on the thermomechanical tangential stress and material properties in the zone of the rotor in which the flaw indications lie.

The N-value of an indication takes into consideration the plastic zone around the indication. FRAC computes the N-value for zones (Figure 9) within the rotor on the basis of mechanical and thermal stresses. The expression used to calculate N-value is

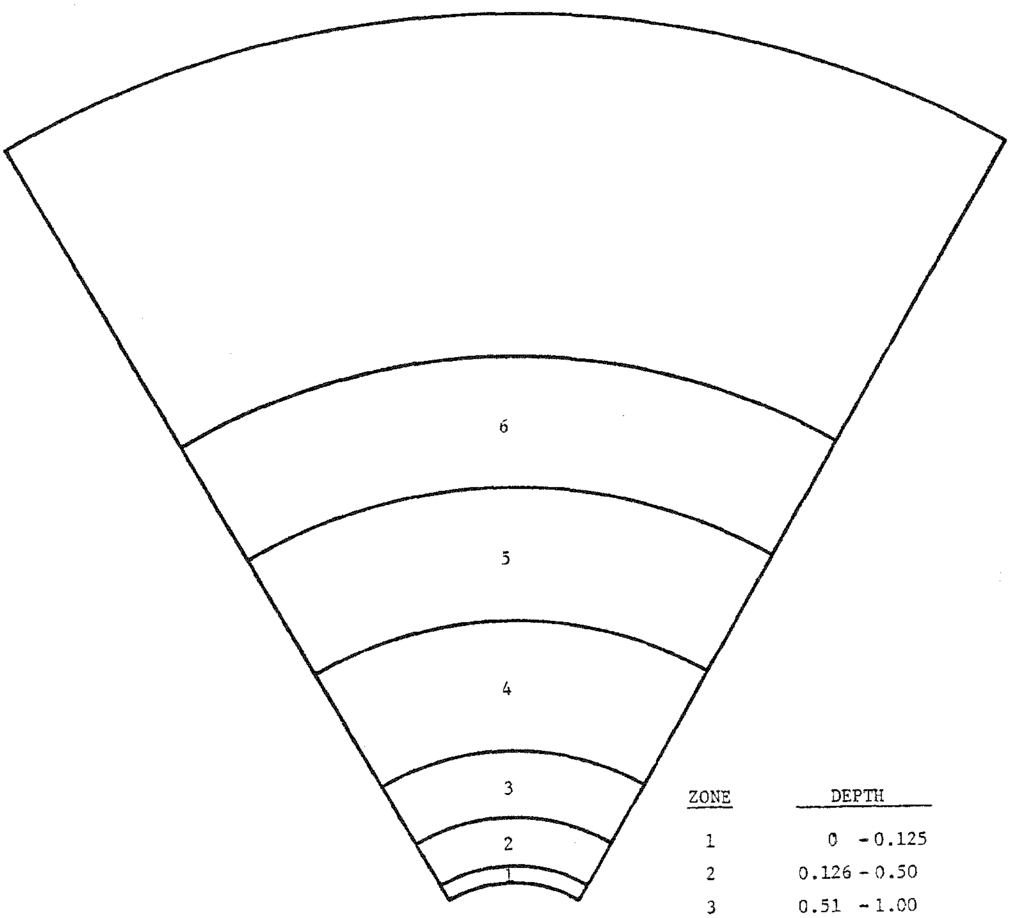
$$\text{N-value} = \left(\frac{\sigma_y}{\sigma_y - \sigma_{os}} \right)^{1/2}$$

where

σ_y = the material yield stress, (ksi), and
σ_{os} = the thermomechanical overspeed stress, (ksi).

There are six radial zones within each axial zone. The length of each axial zone is determined by a twenty percent change in the tangential stresses along the bore. FRAC will support twenty axial zones.

The more highly stressed radial-axial directions around a flaw are taken into consideration by defining a cone-of-exclusion, see Figure 10. This cone has its vertex on the flaw indication and its axis normal to the r-z plane passing through the flaw indication. The vertex angle of the cone-of-exclusion is provided by the user and is a critical input needed to limit the extent of out-of-plane linkage. If a flaw indication is within the search length, it is determined whether or not the point lies outside the cone-of-exclusion.



ZONE	DEPTH
1	0 - 0.125
2	0.126 - 0.50
3	0.51 - 1.00
4	1.01 - 2.00
5	2.01 - 3.00
6	3.01 - 4.00

FIGURE 9. SCHEMATIC FOR ROTOR R-θ CROSS-SECTION: THE SIX FIXED RADIAL ZONES

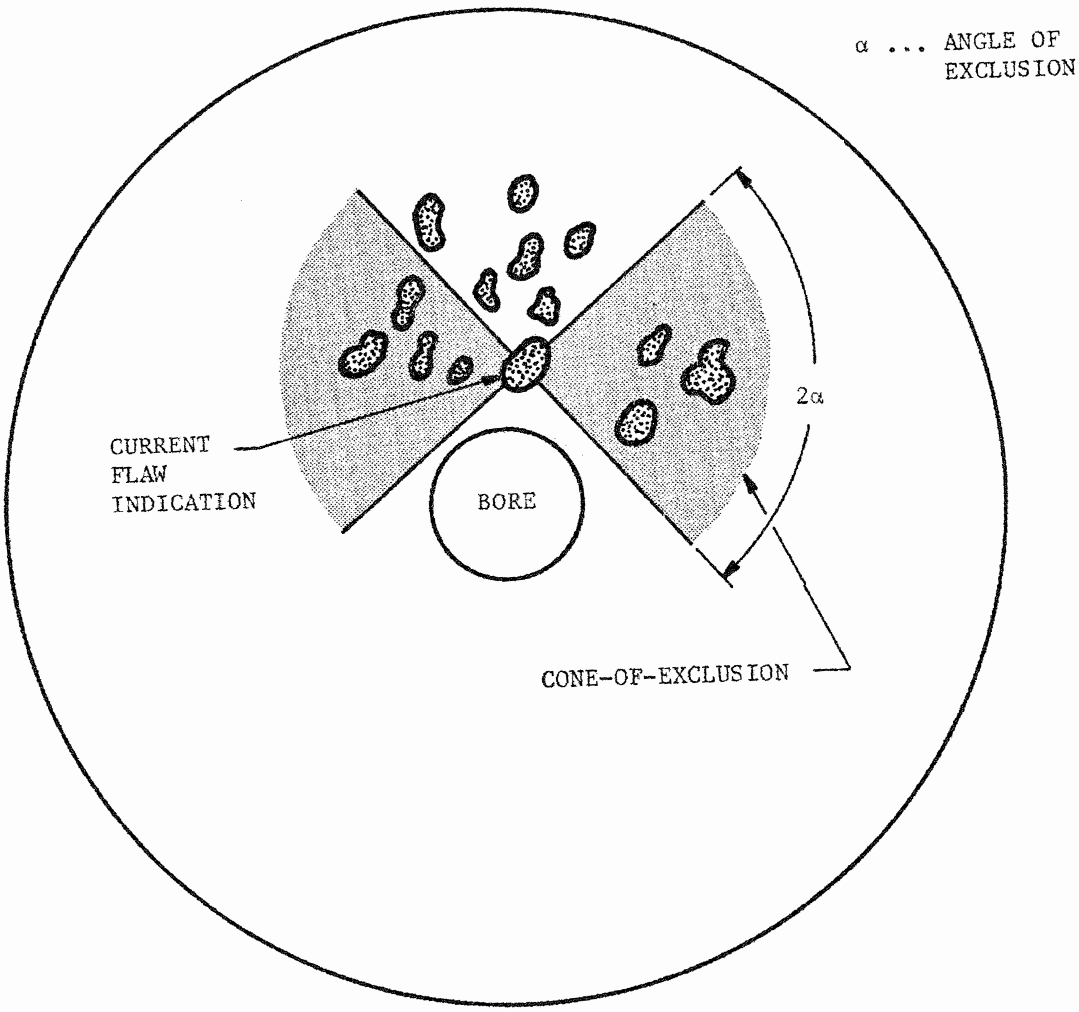


FIGURE 10. SCHEMATIC FOR ROTOR R-θ CROSS SECTION: THE CONE-OF-EXCLUSION

When it is determined that all points beyond the current cluster are beyond the search length, the cluster sequence number and the number of points in the cluster are printed. This header data is followed by the parameters of the points within the cluster. The same information is also written to a file for later use by ELIPSE.

3.2.6.2 Description of ELIPSE Routine

A topological study of the spatial distribution of the flaw indications within a cluster is performed by ELIPSE. ELIPSE collapses the points within the cluster of flaw indications onto an r-z plane containing the centroid of the cluster, and provides the following information:

- The x, y, z coordinates of the centroid;
- Orientation of the ellipse relative to the rotor axis;
- Length of the axes of the ellipse;
- Defect area fraction of the ellipse; and
- Flaw code indicating the aspect ratio of the ellipse.

ELIPSE circumscribes the collapsed cluster by an ellipse and determines the orientation and aspect ratio of the ellipse. A matrix of the x, y, and z coordinates of the indications in the cluster is created, and the eigenvectors and eigenvalues of the matrix are determined by a principal axis transformation. Once the best symmetrical placement of the indications is determined by the principal axis, the coordinates of the cluster indications are transformed from the rotor-system to the cluster-system. The standard deviation of the cluster coordinates about the cluster-axis is used to determine the radii of the ellipse. Once the radii are determined, the area of the enclosing ellipse is calculated; the area of the indications is summed, and the defect area fraction is calculated. FRAC then treats the ellipse as an initial sharp crack.

4.0 LIST OF REFERENCES

1. Swanson, J. A., and Desalvo, G. J., "ANSYS Engineering Analysis System User's Manual," Swanson Analysis Systems, Inc., 1975.

2. Martin, H. M., "Article on Leakage Through Dummy Pistons," Engineering, January 10, 1908 and January 3, 1919.

3. Kapinos, V. M., and Gura, L. A., "Investigation of Heat Transfer in Labyrinth Glands of Static Models," Teploenergetika, No. 11, Volume 17, 1970, pp. 38-41.

4. Brandt, D. E., "The Development of a Turbine Wheel Design Criteria Based Upon Fracture Mechanics," Journal of Engineering for Power, October 1971, p. 411.

5. Pennick, H. G., Cook, T. S., and Wells, C. H., et. al., "STRAP: Steam Turbine Rotor Analysis Programs," Part III, Southwest Research Institute, June 1980.

6. Pennick, H. G., Cook, T. S., and Wells, C. H., et. al., "STRAP: Steam Turbine Rotor Analysis Programs," Part II, Southwest Research Institute, June 1980.

7. Furillo, F. T., Purushothaman, S., and Tien, J. K., "Understanding the Larson-Miller Parameter," Scripta Metallurgica, Vol. 11, Pergamon Press, Inc., 1977, pp. 493-496.

8. Wang, Chi-Teh, Applied Elasticity, McGraw-Hill, 1953, pp. 54-56.

9. Zienkiewicz, O. C., The Finite Element Method in Engineering Science, McGraw-Hill, 1971, pp. 168-169.

10. Weisz, D. A., "Analysis of TVA Gallitin #2 Rotor Burst, Part II Mechanical Analysis," 1976 ASME-MPC Symposium on Creep-Fatigue Interaction, R. M. Curran, Editor, G00113, p. 25.

APPENDIX A

EVALUATION OF THE ACCURACY OF THE PPMESH/ANSYS ANALYSIS

APPENDIX A - EVALUATION OF THE ACCURACY OF THE PPMESH/ANSYS ANALYSIS

PPMESH has been designed to generate a finite element mesh that produces reasonable accuracy at reasonable cost. Various checks have been performed to evaluate the accuracy of the stresses generated by the present PPMESH/ANSYS programs. One specific check was on the Gallatin station rotor, which has been analyzed independently by Westinghouse. The PPMESH generation for the Gallatin rotor is shown in Figure A-1. The mesh for which a closed-form analytical solution was available is shown in Figure A-2. This mesh closely approximates the mesh of the last five elements near the bore below row seven in Figure A-1.

The results of the finite element check analysis are compared in Table A-1 to the solution given by Wang (1953)[8] for a thick cylinder under uniform load. These tabulated results show good agreement between the finite element analysis and the exact solution, with the largest disagreement being about 7.4%. As can be seen from Figure A-3, this difference occurs about 1.18 inches out radially from the bore surface. This may be viewed as an inaccuracy in the element or, in fact, as an indication that the element is trying to represent the concentration in a refined way by Zienkiewicz, 1971.[9] This point may be better understood by noting that the points at R = 2.65, 3.17, and 3.68 are all Gauss points contained within the same element. The accuracy of the element along the linear portions of the curve near the concentration may be explained by the linear nature of the element. By the same token, this reasoning will serve to explain the inaccuracies near the curved portion in Figure A-3. In order to better define the curved region, a more refined mesh is needed.

Although σ_θ is the stress of primary interest, a comparison between θ_r and the exact solution is included in Table A-1 and shown in Figure A-4.

As can be seen in Figures A-3 and A-4, the PPMESH-ANSYS combination does a good job of predicting the stresses near the bore of this thick cylinder loaded by 1000 psi of external pressure.

Upon comparing the results of the thermomechanical stress analysis of the Gallatin rotor performed using the PPMESH and the published results of the same analysis performed by Westinghouse (Weisz, 1976),[10] it is very evident that the two independent analyses give very close results. Figure A-1 is a plot of stress contours from the PPMESH analysis. Superimposed on this plot are the two Westinghouse results for 30 ksi and 60 ksi. The 30 ksi contours follow one another almost exactly except that as the contours pass beneath blade row nine, the PPMESH contour tends to decrease slightly more rapidly. For the 60 ksi stress, the PPMESH tends to encompass a smaller axial dimension than the Westinghouse stress. The contour determined by PPMESH starts at the end of blade row six and terminates about the middle of blade row eight, while the Westinghouse computed stress contour initiates in the middle between blade rows six and seven and ends at the beginning of blade row nine. The difference in height between these two 60 ksi contours leads one to believe that the PPMESH maximum bore stress may be one or two ksi higher than the maximum found in the Westinghouse analysis.

In conclusion, the PPMESH generated for the Gallatin rotor can be considered to produce accurate stress results. A more refined mesh will give greater accuracy; but, if properly done, that refinement would lead to an unnecessary increase in computer cost.

TABLE A-1

TABLE COMPARING EXACT STRESS VALUES TO THOSE GENERATED BY FINITE ELEMENT METHOD

	Finite Element Model (ksi)		Exact Solution (ksi)		% Difference	
R	σ_θ	σ_θ	σ_θ	θ_r	σ_θ	σ_r
2.65	-1.926	-0.133	-1.944	-0.136	-0.9	- 2.0
3.17	-1.562	-0.306	-1.672	-0.408	-6.6	-25.0
3.68	-1.397	-0.564	-1.509	-0.571	-7.4	- 1.2
3.98	-1.408	-0.621	-1.441	-0.639	-2.3	- 2.8
4.50	-1.324	-0.688	-1.353	-0.726	-2.1	- 5.2
5.02	-1.283	-0.772	-1.292	-0.788	-0.7	- 2.0
5.32	-1.267	-0.800	-1.264	-0.816	0.2	- 2.0
5.83	-1.221	-0.832	-1.227	-0.853	-0.5	- 2.5
6.35	-1.193	-0.871	-1.197	-0.882	-0.3	- 1.2
7.30	-1.160	-0.908	-1.159	-0.921	-0.09	- 1.4
8.85	-1.124	-0.952	-1.121	-0.959	0.3	- 0.7
11.90	-1.085	-0.993	-1.085	-0.995	0	- 0.2

$$\sigma_\theta = -6348.400\ (1/R^2) - 1040.312$$

$$\theta_r = 6348.400\ (1/R^2) - 1040.312$$

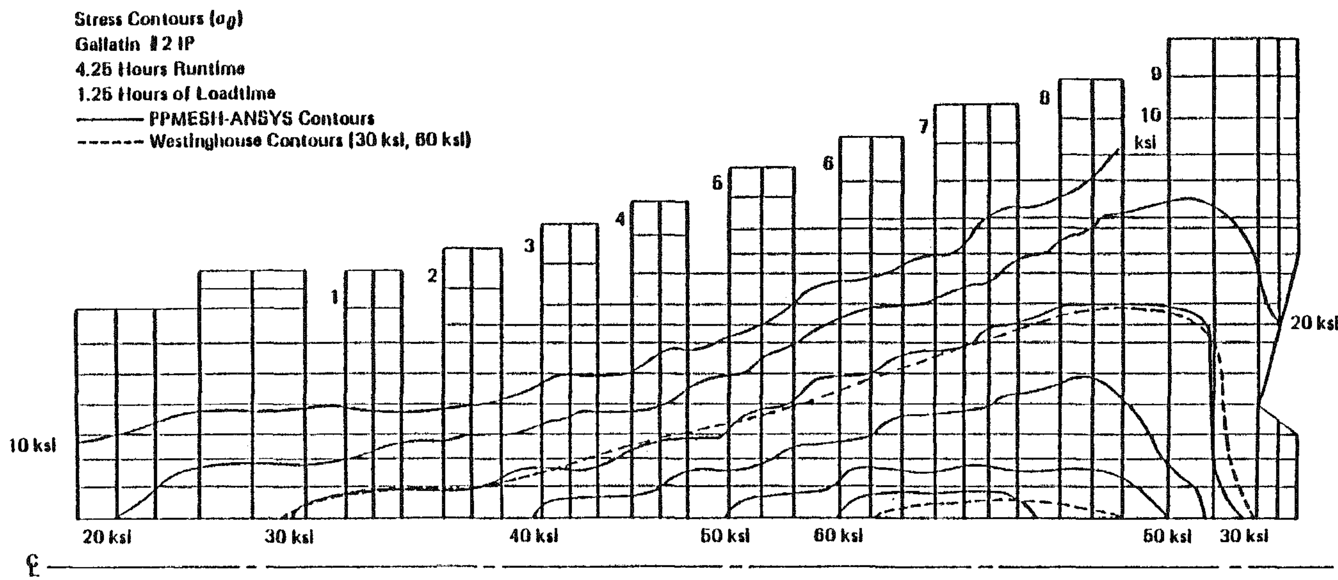


FIGURE A-1. SCHEMATIC OF GALLATIN MESH AND STRESS CONTOURS

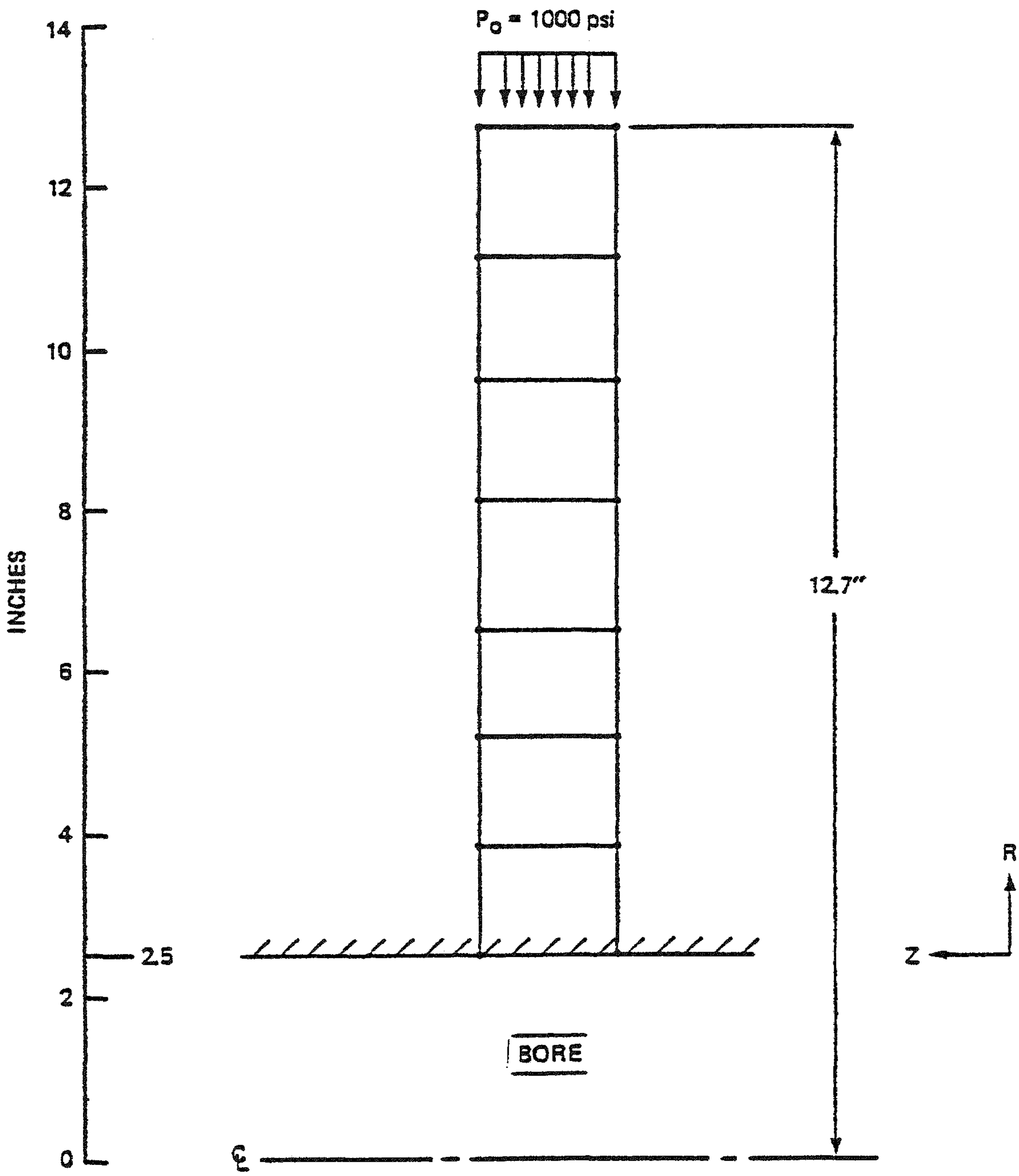


FIGURE A-2. FINITE ELEMENT IDEALIZATION OF TEST MESH

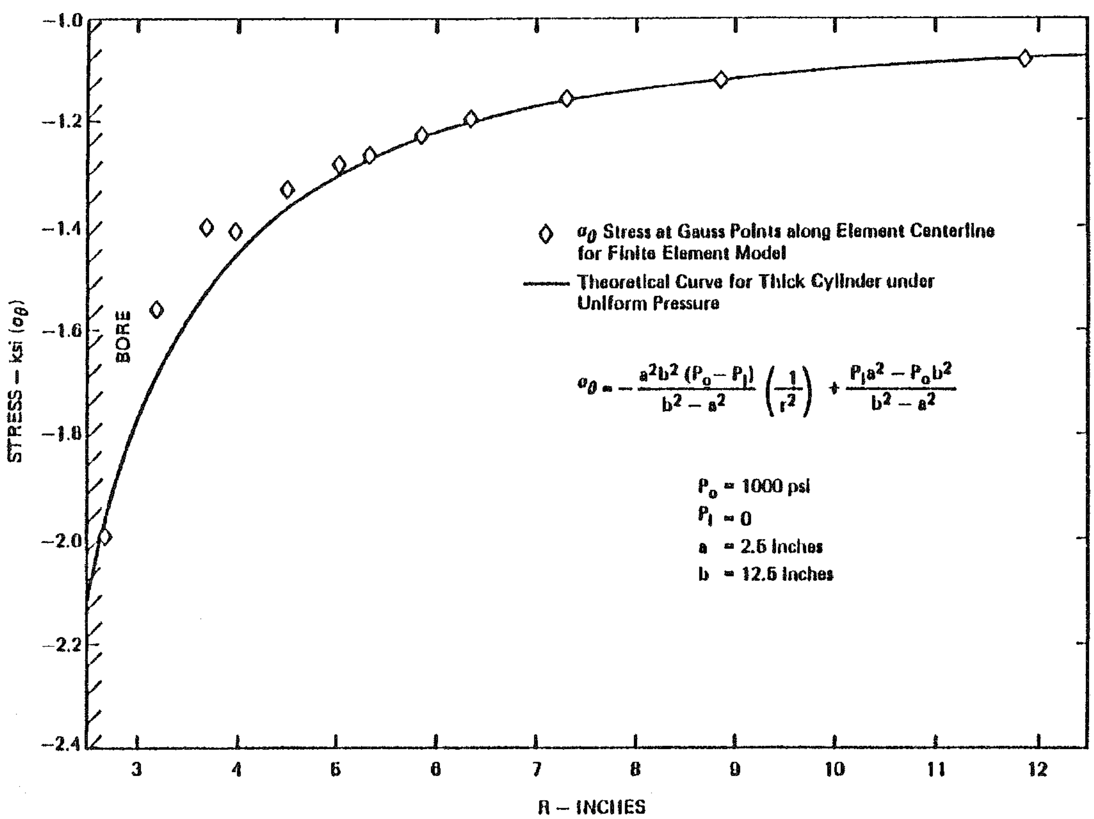


FIGURE A-3. COMPARISON OF EXACT HOOP STRESS AND FINITE ELEMENT HOOP STRESS FROM TEST CASE

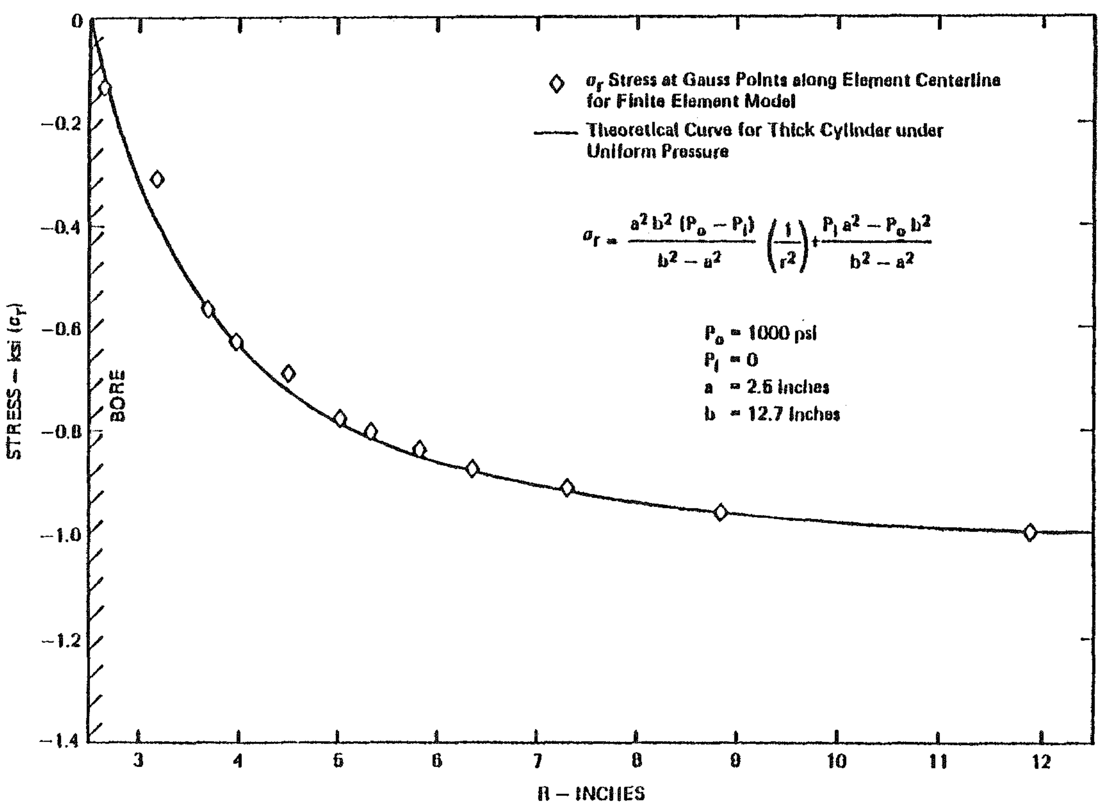


FIGURE A-4. COMPARISON OF EXACT STRESS AND FINITE ELEMENT RADIAL STRESS FROM TEST CASE