LAWRENCE LIVERMORE LABORATORY

University of California, Livermore, California 94550

NOTICE

This report was prepared as an account of work sponsored by the United States Government. Neither the United States nor the United States Energy Research and Development Administration, nor any of their employees, nor any of their contractors, subcontractors, or their employees, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness or usefulness of any information, apparatus, product or process disclosed, or represents that its use would not infringe privately owned rights.

UCRL-52281

**THE DESIGN AND EVALUATION OF A TWO-PHASE TURBINE
FOR LOW QUALITY STEAM-WATER MIXTURES**

William Jabez Comfort III
(D. Engr. Thesis)

May 16, 1977

MASTER

ep

DISTRIBUTION OF THIS DOCUMENT IS UNLIMITED

DISCLAIMER

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency Thereof, nor any of their employees, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof.

DISCLAIMER

Portions of this document may be illegible in electronic image products. Images are produced from the best available original document.

The Design and Evaluation of a Two-Phase Turbine for
Low Quality Steam-Water Mixtures

By

WILLIAM JABEZ COMFORT III
B.S. (San Jose State University) 1963
M.Eng. (University of California, Berkeley) 1964

DISSERTATION

Submitted in partial satisfaction of the requirements for the degree of

DOCTOR OF ENGINEERING

in

Engineering

in the

GRADUATE DIVISION

of the

UNIVERSITY OF CALIFORNIA

DAVIS

Approved:

Charles W. Beadle
Clyde D. Cunningham
Warren H. Dietz

Committee in Charge

Deposited in the University Library.
Date Librarian

This work is dedicated
in memory of my father,

William J. Comfort Jr.
1907-1975

TABLE OF CONTENTS

	page
Abstract	v
Introduction	1
Design Conditions and Introduction to Two-Phase Fluid Dynamics . .	4
Selection of Expander Type	12
Dynamic Blade Test Apparatus	23
Basic Geometry and Sizing	26
Blade Design Considerations	29
Other Features of the Machine	37
Selection of Fabrication Techniques and Materials	40
Major Aspects of Fabrication of the Rotor	41
Structural Analysis	49
Static Cascade Design Considerations and Description	59
Numerical Performance Modeling	61
Discussion of the Model	70
Static Cascade Testing and Analysis	73
Dynamic Blade Apparatus Testing and Analysis	80
Extrapolations Using the Numerical Performance Model	98
Recommendations for Future Effort	108
Conclusion	109
Acknowledgements	111
References	113
Appendix 1: Nomenclature	117

	page
Appendix 2: The Hero or Radial Outflow Turbine	121
Discussion	138
Appendix 3: Structural Considerations	140
Finite Element Modeling of the Rotor	140
Blade Vibration	146
Radial Blade Displacements	151
Structural Damping Measurements	152
Appendix 4: Nozzle Characterization	154
Appendix 5: Turbine Performance Model	164
Coriolis Pumping Including Liquid Film Calculations	182
Losses Associated with Low Velocity Liquid Exiting the Nozzle	190
Disk Friction and Windage, Partial Admission, and Tip Clearance Losses	191
Appendix 6: Two-Phase Shock Demonstration	195
Appendix 7: Description of Supporting Experimental Apparatus and Instrumentation	199

Abstract

A new two-phase turbine has been designed, and built for testing in the laboratory, using a low quality steam-water mixture as a working fluid. This expander has been successfully tested in the laboratory as a part of the Lawrence Livermore Laboratory geothermal energy development program. The measured performance compares well with performance predictions of a numerical model of the expander.

This report describes: 1) the details of the selection of the type of expander, 2) the design of an experimental expander for use in a clean two-phase flow laboratory experiment, 3) the development of a numerical model for performance analysis and extrapolations, 4) experiments including static cascade performance with two-phase fluid, disk friction and windage measurements, and two-phase performance measurements of the experimental expander, 5) comparisons of the numerical model and experimental results, and 6) the prediction of the performance of an advanced design, indicating how performance improvements can be achieved.

An engine efficiency of 23 percent for a single-nozzle test was measured. Full admission performance, based upon the numerical model and achievable nozzle thrust coefficients indicate that an engine efficiency of between 38 and 48 percent can be realized with present technology. If maximum liquid removal loss is assumed, this performance range is predicted to be 38 to 41 percent. Droplet size

reduction and the development and implementation of enhanced two-phase flow analysis techniques should make it possible to achieve the research goal of 70 percent engine efficiency.

Introduction

The fluid for a majority of geothermal resources is naturally two-phase at the wellhead unless special pumping techniques are employed. The fluid mixture is not pure water but includes dissolved minerals and gases constituting as much as 30 percent by weight. In addition there are particles such as sand or small pieces of scale or other debris. Conventional two-phase geothermal power plants in Mexico and in New Zealand separate the vapor from the remainder of the fluid, exploiting only the vapor fraction in standard steam turbines. Since the liquid is not used, a large amount of available thermal energy is wasted.

It has been proposed [1], [2] that a single two-phase expander can provide the most efficient utilization of two-phase geothermal resources. A thermodynamic process requiring the simultaneous expansion of both the liquid and vapor components of a two-phase steam-water mixture was shown to produce work equivalent to the available energy in the ideal case. The process of expanding the two-phase mixture directly from well-head to condenser fluid states is called the Total Flow process.

In addition to geothermal applications, the Total Flow process should be highly useful for waste heat energy recovery. With increased awareness of the need for energy conservation, two-phase expanders should be seriously considered for conversion of the thermal energy in hot fluids associated with industrial processing. Two-phase expanders may also find application in bottoming cycles for large conventional

and nuclear power plants and in solar energy applications where energy densities in the working fluid are low.

The research goal, established by Lawrence Livermore Laboratory (LLL) management, for the two-phase Total Flow expander engine efficiency is 70 percent. The three major objectives of the present effort are: 1) the selection of an appropriate expander for low quality two-phase flow, 2) the evaluation of the selected expander experimentally and analytically, and 3) the use of the insight gained through the evaluation process to indicate how a future expander can be designed to achieve enhanced conversion efficiency and/or to meet the research goal.

The steps which have made it possible to achieve these objectives will be described in this report. These steps include: 1) the details of the selection of the type of expander, 2) the design of an experimental expander for use in a clean two-phase flow laboratory experiment, 3) the development of a numerical model for performance analysis and extrapolations, 4) experiments including static cascade performance with two-phase fluid, disk friction and windage measurements, and two-phase performance measurements of the experimental expander, 5) comparisons of the numerical model and experimental results, and 6) the prediction of the performance of an advanced design, indicating how performance improvements can be achieved.

The experimental devices designed to meet the above objectives should be viewed as laboratory tools used to address the questions of

two-phase flow through a single expander. The expander is not a final design for a field application nor a machine of optimized performance. The investigatory nature of the device allows for latitude in material selection and life expectancy requirements which would not be possible for a field design. All testing reported herein was accomplished with clean steam-water flow.

Most turbomachinery is built to operate with either vapor or liquid, but not with both simultaneously. Condensing steam turbines do include small fractions of liquid in the vapor working fluid, but the liquid fraction is looked upon as a necessary evil—not as a potential to do good. Conversion of the potential energy contained in both phases to useful power is unique to the fluid conditions and to the turbomachinery requirements presented herein. As will be explained in more detail later, the extreme differences in the fluid mechanic behavior of the liquid and the vapor phases in conversion machinery make traditional fluid machinery modeling laws almost useless.

A wealth of literature exists concerning expanders for saturated steam, where steam quality may be as low as 90 percent, see for example: [3],[4],[5]. Elliott and Hays [6] have designed and tested several turbines and separators using two-component (nitrogen-water) two-phase fluids. Their work has, until recently, been directed towards expanders designed to replace magnetohydrodynamic generators and towards using very low fractions of vapor-to-liquid mixtures to effectively increase the density of the working fluid for two-phase expanders. The high effective density of the working fluid is

advantageous in allowing lower operating speeds for conversion machinery while maintaining high ratios of power producing capability to the size of the expander. Motive fluid mixtures in these studies have been less than 0.03 vapor mass flow rate fraction. Such mixtures can lead to significant liquid layers in the passages of the conversion machine. While some geothermal processes can involve such vapor mass flow rate ratios, the problem being addressed herein is one, as will be discussed in the next section, in which liquid layers in the passages of the expander are minute or non-existent. The single-component steam-water mixtures, used to represent the geothermal fluid, also results in additional considerations which are not present in two-component investigations. Indeed, the design and evaluation of an expander for low quality two-phase mixtures of steam and water is unique.

The highly complex nature of two-phase flow and the implications of the particular fluid conditions to which this project relates make it crucial to first introduce some basic two-phase fluid behavior. Understanding of the behavior of the liquid and vapor mixture in an expander requires an appreciation of the peculiarities of the working fluid as well as an insight into conventional turbomachinery design.

Design Conditions and Introduction to Two-Phase Fluid Dynamics

The operating conditions, Table 1, for which a two-phase expander was to be designed were based upon the conditions selected in [1]. Though

other possible initial and final states were considered during the conceptual and developmental process, the listed conditions prevailed. The well-head condition continued to represent the fluid state corresponding to the high-temperature, high-salinity geothermal wells of the Salton Sea area in Southern California which was and is of major interest to the LLL geothermal program. The exit pressure was based upon the minimum sink temperature available on the desert on a warm day, thus providing maximum energy extraction per well. No systems analyses have yet been made for the selection of exit pressure based upon such variables as costs and/or dissolved gas evolution.

Table 1
Initial design conditions

	Inlet	Exhaust
Pressure	2.482 MPa (360 psia)	0.0124 MPa (1.8 psia)
Temperature	223.5°C (434.4°F)	50.1°C (122.2°F)
Vapor mass fraction	0.189	0.35
Vapor volume fraction	0.940	0.99984
Enthalpy	1.307 MJ/kg (562.4 Btu/lbm)	1.043 MJ/kg (448.7 Btu/lbm)
Isentropic homogeneous nozzle exhaust velocity	—	727 m/s (2385.8 ft/s)

There are several items to be noted concerning the design operating conditions. Though the pressure ratio is quite large, the isentropic enthalpy change is not excessive. This is in distinct contrast to a pure vapor working fluid where pressure ratios of this magnitude would produce unreasonably large enthalpy changes. A major portion of the

work output of an isentropic process between the design states is the result of expansion below atmospheric pressure. Also, expansion below atmospheric pressure leads to large volume expansions of the fluid. Though the liquid mass fraction is large throughout the process, the liquid volume fraction remains very small.

As may be deduced, Table 1, from the large void fraction, which implies very small liquid volume fraction, the working fluid may be characterized as droplets in a vapor continuum for all flow conditions of practical significance. This characterization does not preclude separation of high velocity two-phase mixtures resulting from body forces occurring, for example, in flow of a mixture of droplets and vapor through a curved channel. Another extremely important consideration is the amount of liquid available to form a layer on flow channel walls. The high void fractions imply that liquid layers may be very thin, perhaps forming rivulets, even though the mass fraction of water is quite appreciable.

Another interesting phenomenon encountered in two-phase mixtures is that of choking. As with a pure vapor, choking occurs at the minimum area of a converging-diverging duct. If the droplets are very small (much less than 1 micron), the mixture behaves as a homogeneous continuum, which is consistent with pure vapor behavior. The choking velocity is defined in equation (1)

$$c = \sqrt{\left. \frac{\partial p}{\partial \rho} \right|_{is}} \quad (1)$$

where p is the pressure and ρ is the mixture density of the fluid. The magnitude of this two-phase mixture choking velocity is lower, however, than either the choking velocity for the pure vapor or the pure liquid.

When droplets are large enough so that the mixture does not behave as a continuum, the coupling between the droplets and the vapor is weak, which leads to slip (S , defined here as the droplet velocity divided by the vapor velocity) between phases. When slip is present, the continuity averaged velocity (so named because it appears in the continuity equation of the two-phase mixture equations in conservative form) is related to the thermodynamically defined choking velocity by the approximation shown in (2)

$$[XV_V + (1-X)V_L]_{\text{critical}} \approx \sqrt{\frac{\partial p}{\partial \rho}}_{\text{is}} \quad (2)$$

where X is the vapor mass fraction, V_V is the vapor velocity, and V_L is the droplet velocity. The true choking point, however, can only be found through calculations in which the vapor and droplets are fully coupled. Such calculations can be performed with a computer code, LD2FAZ [7], which has been developed for the modeling of quasi-one-dimensional two-phase fluid flows including normal shock waves. In this calculational model, the liquid and vapor phases can have different velocities, but it is assumed that the droplets are small enough so that thermal equilibrium is maintained.

The vapor mass fraction, X , is not the same as the vapor mass flow-rate ratio, R , unless the mixture is in kinetic equilibrium. The vapor mass flow rate ratio is the same as the quality as defined by the equation of state. The relationship between the two quantities is given in (3).

$$R = \frac{X}{X + (1 - X)S} \quad (3)$$

As shown, the relationship is a simple function of the slip ratio.

The change in the continuum choking velocity both with respect to pressure (at the approximate entropy of the design expansion process) and with respect to vapor mass flow-rate ratio (at the minimum pressure of the design expansion process) is shown in Figures 1 and 2, respectively. These results are accurate when the mixture behaves as a continuum and thermal equilibrium exists between phases. For most processes under consideration, this is a reasonable approximation and will be employed unless otherwise noted.

Once the two-phase mixture choking velocity has been exceeded, the mixture behaves like a supersonic fluid, i.e. acceleration occurs with negative pressure gradient and positive area change. When droplets are relatively large, the vapor can still carry information at the vapor sonic velocity. As the droplets become smaller, dense gas behavior is approached so that the velocity with which information travels approaches the two-phase mixture choking velocity.

Fig. 1. Choking velocity versus pressure at constant entropy ($3.22 \text{ kJ/kg } ^\circ\text{C}$) compared with the choking velocities of saturated water and saturated water vapor.

Fig. 2. Two-phase mixture choking velocity versus vapor mass flow rate ratio (quality) at constant pressure (0.012 MPa).

Shock behavior in a two-phase mixture is very difficult to predict because of the wide variety of variables that can affect the location and severity of the phenomenon. Calculations using 1D2FAZ indicate that a steady state normal shock can exist when the two-phase mixture choking velocity is exceeded if the droplet size is small enough. The results of calculations, Appendix 6, show that discontinuous fluid dynamic behavior appears when droplet diameters are approximately 1 micron for thermodynamic conditions similar to those which might be expected in an expansion process based upon Table 1. Less severe flow adjustments result for the same conditions as the droplet size is increased. The critical velocity which must be exceeded for discontinuous flow adjustments to occur becomes successively higher until the vapor sonic velocity is reached. Thus, the two-phase shock can be initiated at flow velocities between the two-phase choking velocity and the vapor sonic velocity depending upon droplet size.

Depending upon the type of expander chosen, fluid velocities can easily exceed the two-phase choking velocity and sometimes also the vapor phase sound velocity. Thus the possibility of two-phase shocks must be addressed when the flow conditions warrant it.

As previously mentioned, body forces must be considered where flow velocities and droplets are relatively large. Nearly any expander one considers, requires sufficiently high velocities and torturous enough flow paths to cause droplets to separate from the vapor. As will be shown later, this can lead to large losses of available energy at interfaces where liquids tend to impact and/or collect.

Complete solution of two-phase fluid dynamics problems requires a coupled approach for the liquid and vapor in which two separate sets of equations for continuity, momentum and energy are written and solved with coupling relations and appropriate equation of state functions [7]. In some cases, one can approximate certain aspects of two-phase fluid flow by assuming that coupling occurs in only one direction. The specific application of this concept, which will be used later, is exemplified by the calculation of droplet trajectories. The vapor phase is assumed to be unaffected by the presence of the droplet, while the droplet is acted upon (through drag) by the flowing vapor. In regions of high void fraction and low pressure gradient, this uncoupled approach has been found to result in a good approximation to the droplet trajectories one would expect in a fully coupled model.

Selection of Expander Type

Table 2 lists the types of expanders which might be considered for use in this application. Positive displacement expanders may be made to operate with reasonable efficiency using a two-phase fluid [8]. House [9], Elliott [10], and others have pointed out that because of the fluid states between which the expander is to operate, a large volume change occurs during the expansion process. This leads to a very large machine or many smaller ones. To preserve simplicity and to minimize the size and/or number of machines, positive displacement devices were eliminated from further consideration as a single stage expander for the Total Flow process. The device does have merit as part of a

multiple stage system and may well find further application in geothermal energy conversion. The remainder of the machines listed are turbines. These will be considered in the following paragraphs.

Table 2
Candidate Expanders

Impulse turbines
Tangential flow
Axial flow
Impulse/reaction turbines
Axial flow
Radial inflow
Radial outflow
Hero's turbine
Positive displacement
Lysholm

Initial guidance, both from the concept [1] and from management stressed the minimization of equipment. When the selection process was initiated, scale control had not been achieved. Minimization of scale formation led to the desire to minimize residence time in the machine because it was believed that scale formation required a finite evolution time. Since minimum residence time also implies a short fluid path, minimization of surface area occurred as a result of this approach, thus reducing the area over which scale could be formed. The separation of liquid and vapor, as a result of body forces, required that multiple stage turbines would necessitate remixing of the phases between stages in order to complete the Total Flow process. Remixing requires additional residence time and loss resulting from heat transfer and work expended in the remixing process. Because the enthalpy change for the design condition was not excessive, and because

of the above considerations, a single stage turbine was chosen. The remaining areas of design choices involve the selection of turbine flow geometry and degree of reaction.

At this point in the development of the turbine, very little was known concerning two-phase fluid mixture behavior. Tests of static nozzles [11] had previously established that nozzles could efficiently accelerate two-phase fluid. Droplet sizes and slip ratios were unknown. Coupled two-phase flow calculation techniques [7], [12], had not been developed. Management was pressing to design and build a concept demonstration device which could be used for laboratory experiments as soon as possible.

The partial reaction turbine, in which a portion of the total enthalpy change occurs across the nozzles (stator) and the remainder occurs across the blades (rotor), seems quite attractive for this application. Such machines can be designed so that the fluid velocity relative to the rotor is small, which is important in order to avoid shock losses and erosion. Erosion is of particular concern in the two-phase environment; the problem being further compounded by grit and particles of scale and the corrosive nature of the brines expected in the field conditions. Several forms of reaction turbines were considered. It was reasoned that in order to attain high efficiencies in a reaction stage, the droplets being accelerated through the rotating passages would have to be so guided as to avoid striking the nozzle walls. High efficiency could be attained if the path through the nozzle were selected by use of coupled two-phase calculations including the body

forces, preferably in three dimensions, and with a detailed knowledge of the distribution of droplet diameters and of the mechanisms of droplet breakup. As previously noted, none of these required ingredients was available during the process of selection of the expander type. It may be argued that droplet breakup is well known. The most frequently used empirical correlation for droplet breakup is the Weber number criteria [13] (see Appendix 4), which is more applicable to two-component than to single-component two-phase flow. Crowe [14] has suggested that in regions of high pressure gradients which are governed by the criteria, the likelihood of thermal non-equilibrium is also highest because slip ratios decrease significantly. This leads to the possibility of local boiling phenomena within the droplets which tends to promote droplet breakup. Though work is presently being done to develop such a model [14], experimental verification is, even now, in its infancy.

Aside from the difficulties of proper two-phase rotor nozzle design, partial reaction designs exhibit several other undesirable characteristics. Scale would be expected to form where negative pressure and temperature changes occurred, i.e. in the rotor passages, leading to possible imbalance and cleaning problems. Partial reaction designs also require rotor sealing (between the case and the rotor). Scale formation in the region of the seals further complicates seal design.

The radial inflow turbine appears to demand three-dimensional fluid flow calculational techniques while axial flow designs appeared to be

approachable in two-dimensions. Because of the relatively large radius change in the radial inflow design, droplets sticking to the walls of the rotating passage would most likely be carried out radially under centripetal acceleration. Thus, the liquid would produce additional flow losses for the passing vapor, and more important, through Coriolis acceleration, would require work to be done by the turbine to remove the droplets. Hence, the radial inflow machine was eliminated from the list of candidates.

Further examination of the concept of the reaction turbine leads one to study the limits of pure impulse (zero reaction) and pure reaction machines. Comparing the idealized single stage turbine on the basis of efficiency versus blade speed (Figure 3), one sees that the greater the degree of reaction, the greater the required blade speed to achieve high efficiency. The pure reaction machine (sometimes called the Hero turbine) is truly radial outflow and does not lend itself well to two-phase flow applications as a Total Flow expander, see Appendix 2. Though actual operating losses significantly reduce the turbine operating blade speed, these losses also significantly reduce its maximum expected performance. Even a high efficiency pure impulse machine would require blade speeds of approximately 350 m/s for the design operating conditions—hence reaction designs for a single stage approach material strength limits. The question of material strength limits does not seem pressing until one remembers that the environment for the actual field application will be quite hostile. Erosion, corrosion, and erosion assisted fatigue considerations require that the basic strength of materials used for the field design must be derated

Fig. 3. Impulse/reaction expander characteristics.

for environmental reasons. As with other reaction designs, the uncertainties of two-phase nozzle design in a rotating frame reduced the probability of success of such a turbine. The radial outflow turbine was not considered further because of structural limits, performance, and the difficulties associated with two-phase flow operation.

The remaining machines to be considered are pure impulse with either tangential or axial flow. As mentioned earlier, it appeared possible to approximate the blade flow field with two-dimensional analysis rather than three. It was concluded that the impulse design was the simplest--and could be approximately analyzed with an uncoupled calculation. Structural requirements are minimized by the impulse blade speed requirements. Losses associated with radial removal of droplets impacting the blade increase as the blade speed squared (see Appendix 5), favoring the impulse design. Accessibility of nozzles for scale control and cleaning is also enhanced in the impulse expander since the nozzles are stationary. In addition, some preliminary nozzle data [15] suggests that nozzle performance may be dependent upon the absolute enthalpy of the entering fluid. If this is proved, there will be an additional reason for choosing an impulse machine for this application. Though it is expected that performance may suffer at very low resource temperatures, there is presently insufficient evidence to indicate that any severe limitation would be thus placed on the impulse design.

Both the tangential (Figure 4) and the axial (Figure 6) flow impulse

machines would have similar blade speeds for optimum performance. Because of the geometry and structural shape of the tangential flow buckets (Figure 4), the axial flow design will exhibit lower stresses in the blade region. Bending moments in the flat portion between buckets resulting from centripetal loading cause this to be a high stress region. Blades for the tangential flow machine must be integral with the hub in all but the most sophisticated of designs. The axial flow blades can be made either integral with the hub or attachable to it.

The fluid path in the tangential flow machine may appear to have advantages over axial turbines with respect to slip. This illusion is the result of constructing vector diagrams for the tangential and axial flow geometry from a viewpoint directly above the bucket. From this view it is possible to conclude that slip does not effect the relative velocity direction in the tangential device. However, when the same vector diagrams are viewed from the axial direction (Figure 5b), one sees that the tangential flow relative velocity direction is also affected by slip. As will be emphasized later, the droplets must be very small in order to pass through the curved blade passages of impulse turbines without striking their surfaces. Because impact with the blade surface and additional efficiency losses associated with removal of the liquid from the blade, the partial turning which occurs (Figure 5a) in the axial blades as a result of slip may be considered an advantage over tangential flow. Also it would be expected that the more guided nature of the axial flow blade channel, compared to the

Fig. 4. Tangential flow bucket geometry.

Fig. 5a. Axial flow liquid and vapor velocity vectors.

Fig. 5b. Axial view of tangential flow liquid and vapor velocity vectors for entrance to bucket.

Fig. 6. General axial flow configuration showing details of the nozzle and blade geometry.

tangential flow blade, would lead to higher efficiency for the axial blading.

Axial flow geometry lends itself well to full admission and to large mass flow per unit diameter of turbine. The blade height may be increased to facilitate large flow rate, within the limitations of acceptable hub to tip reaction change resulting from radial pressure gradients, and liquid pumping losses associated with radial removal of liquid. Further discussion of radial pressure gradients will be included later. The tangential geometry requires additional wheel width for higher mass flow accommodation.

Thus, the axial flow impulse machine emerged from the list of possible candidates as the most promising for initial development. This was not to preclude later consideration of partial reaction designs of various flow geometries when supporting empirical evidence and/or two- or three-dimensional coupled two-phase flow calculation capabilities became available.

Dynamic Blade Test Apparatus Design

In light of the management requirement that results be obtained quickly, the more methodical or basic experiments could not provide sufficiently fast answers. Because nozzle performance had been shown to be acceptable for the fluid conditions of interest, the major turbine performance uncertainty was associated with the rotor. It was

decided that the most expeditious way to explore the many unknowns involved in the turbine design was to build a relatively simple two-phase axial flow impulse machine as first priority, followed by basic experiments, static cascade tests, and additional analytical effort. Management pressure to produce hardware played a strong role in this decision. From purely technical considerations, it could be argued that too little was known about the fluid behavior to adequately design even an impulse turbine. On the other hand, the major objective of management was that of physical demonstration of the concept--because there were many "Doubting Thomases", and several of these controlled the funding. Another important management goal was also involved--that of creating a focus and a physical objective for the technical staff.

The apparatus which will now be described (Figure 6) is purposely called a dynamic blade test apparatus rather than a turbine. A turbine implies a prime mover capable of operating in the intended conditions. The device under consideration is a laboratory tool. Its intended life span is relatively short, only long enough to obtain the required experimental data. A lifespan of only a few hours is sufficient. The experimental working fluid contains no dissolved solids. Material selection considerations are thus significantly relaxed compared to the requirements for a field design. The primary purpose of the apparatus is to provide experimental data leading to better understanding of single-component two-phase flow in turbine blades, and the associated loss mechanisms. If experimental results obtained under these rather idealized and controlled conditions were unencouraging, then it would

not be prudent to proceed to a field design where additional variables would be introduced.

Before describing the rationale used to determine the size and geometrical relationships for the apparatus, it is necessary to discuss the applicability of similarity laws as a basis for turbomachine design and performance prediction for two-phase working fluids. It has been found that, for single phase turbomachines, characterization graphs [16], [17] employing the dimensionless parameters of specific speed and specific diameter can be very useful to the designer. These graphs, based upon similarity laws, have long been used for sizing and performance estimating of conventional fluid machinery. It would seem that a similarity approach could also be used for the machine in question.

The concept of similarity, particularly in turbine design, involves similarity of entering and exiting velocity triangles between "similar" machines. If one examines the entering and exiting velocity vector triangles for a single phase fluid and for a homogenous mixture behaving as a continuum, no differences will be evident. However, if droplets in the mixture now become large enough so that slip occurs in the fluid, two sets (assuming for the present droplets of all the same size and velocity) of entering and exiting velocity triangles emerge: one set for the liquid droplets, and one set for the vapor. The comparison of these two cases was shown in Figure 5a. If one next considers wall interaction with the droplet phase such that some droplets do not emerge at the exit but leave radially, further validity

is removed from the basic single phase similarity relationships. The difference in viscosity between the two phases results in large differences in the Reynolds numbers for liquid and vapor, further invalidating similarity. As may be seen, single phase similarity relationships break down rapidly in two-phase turbomachines. Because of these considerations, similarity relationships were not used in designing the two-phase expander described herein.

Basic Geometry and Sizing

The basic size of the machine was dictated by the nozzle geometry with some thought given to sizing a field demonstration turbine. Nozzle designs previously tested had had throats of approximately 7.5 mm diameter. It was originally thought that the geothermal test facility [18] could supply flow rates with the required temperature and quality for only a single nozzle of this size but later it was found that two nozzles of this size could be accommodated. Scale control had not been established [19] prior to this design. Hence, an additional consideration was the minimum size throat one would feel comfortable with for a field design. No experience was available concerning performance as a function of throat size (or geometry). It was decided that smaller nozzles were not desirable in anticipation of the later field designs. Thus, it was decided to use a nozzle size consistent with previous experience.

Annular nozzle exits, as are frequently used in standard vapor

turbines, could not be reliably designed. Circular cross-section nozzles, as had been tested, would provide poor use of available flow introduction area in an ultimate full admission design. It was decided, that extension to the rectangular nozzle cross-sections should be accomplished for the laboratory test unit in order to begin to understand the complexities which might be encountered in the field. As in previous experimental nozzle designs, the turbine nozzle was designed [20] with a straight axis to avoid body force and supersonic flow complexiities in curved passages.

Using basic velocity vector relations for nozzle and blade angles, it was initially decided that a nominal nozzle angle might be 15 degrees with respect to the plane of the rotor, and that the blades would then be 28 degrees with respect to the plane of the rotor (Figure 6). The nozzle angle was later modified based on two-phase turbine performance estimates using a numerical model (see Appendix 5) which was not available at the time the initial designs were required.

Operating conditions, previously described, for the nozzle dictated an exit area. The angle of inclination of the nozzle then dictated the "nozzle footprint" (Figure 6) in the plane of the rotor. A square footprint leads to acceptable nozzle cross-section aspect ratio. When the footprint becomes rectangular, with the long side perpendicular to the radial direction, the flow conditions at the corners of the rectangle become increasingly questionable because the flow exiting the nozzle exhibits a growing radial component with respect to the blades. In the other extreme, when the short side is perpendicular to the

radial direction, unnecessarily long blades with radial reaction complications, and unreasonable nozzle cross-section aspect ratios result. These considerations, coupled with conventional impulse blade height-to-rotor radius ratios (H/r_o) then lead to establishment of the rotor geometry. As will be discussed later, conventional vapor H/r_o ratio restrictions may be more severe than necessary for two-phase turbine design.

The final geometry chosen also resulted in a full admission power output capability in excess of 1 Mw which was established by management as the field demonstration size. Again, the laboratory machine described here would not be taken to the field, but it was reasonable to establish its geometry such that a similar brine tolerant rotor for field testing could be built without gross changes.

The attitude of the rotational axis of the rotor was chosen to be vertical because of two basic two-phase fluid considerations. First, in a full admission geometry, which the ultimate design would have, uniform distribution of both phases is essential in order for nozzle performance to be uniform and efficient. Since the nominal design conditions introduce relatively low velocity two-phase fluid mixtures to the turbine inlet, gravitational effects will be important in determining flow distribution from nozzle to nozzle. In the vertical axis configuration, it is possible to design a two-phase distribution system so that gravity acts to maintain symmetry of the flow. Thus it provides the most advantageous conditions for uniform distribution. Second, liquid collecting in the space between the rotor and the

turbine housing is uniformly removed by gravity in the vertical shaft configuration to minimize the possibility of additional liquid/rotor efficiency losses due to liquid entrainment or liquid film bridging between the rotor and the rotor housing.

Blade Design Considerations

The inability to measure droplet size, and unavailability of developed tools to make possible detailed fluid dynamic design of the blades for the laboratory test unit, made it necessary to resort to crude techniques for blade design. In order to appreciate some of the complexities involved, it is appropriate to review two-phase fluid behavior previously presented for the particular application of blade design.

Design fluid velocities relative to the blades were estimated to be between 350 and 500 m/s, depending upon the rotor speed. If one calculates droplet trajectories (calculation of such trajectories is discussed in Appendix 5), it may be seen that (Figures 2 and 7) the two-phase choking velocity may be exceeded, even for velocities which are subsonic with respect to the vapor, leading to supersonic fluid behavior in the mixture. Near the blade surface, where droplets are impacting, particularly low quality mixtures may occur, accentuating the supersonic mixture behavior. In the region of higher quality, either subsonic or supersonic behavior may result depending upon the local conditions. Exceeding the two-phase mixture choking velocity

Fig. 7. Subsonic/supersonic fluid mixture behavior in blade passage.

also leads to the possibility of discontinuous flow adjustments, depending upon droplet size, as described previously. The vapor sonic velocity can also be exceeded. For the cases where the flow is supersonic relative to the vapor velocity, shocks can occur, independent of droplet size.

The importance of understanding the droplet behavior cannot be over-emphasized. Because of the large mass fraction associated with the droplets, even though they are traveling at lower velocity than the vapor, it can be estimated that the kinetic energy associated with the liquid is approximately equal to that of the vapor as the mixture exits the nozzle.

Because of the high void fraction, droplets striking the blade may encounter nearly dry surfaces. Continuous liquid films appear to be unlikely, while rivulets formed from contiguous droplet impacts are credible.

Sharp leading edges were chosen for the blades in order to minimize shock losses associated with the leading edge. The practicality of a sharp edge in an erosive environment remains to be proved, however initial experiments [21] look promising. The decision to ignore the implications of erosion for the first blade profile was made in order to create an experimental device which would specifically address the loss mechanisms associated with the aspects of two-phase fluid behavior which could not be controlled by design. Thus sharp edges were desirable to minimize the number of geometrical variables which might

result in poor performance for other reasons, i.e. supersonic relative vapor velocity and resultant shock losses. Later when more basic experiments can be performed, it is anticipated that trade-offs between erosion and shock losses will have to be investigated.

The blade width was based upon what was considered a manufacturable size with emphasis toward a larger radius of curvature. Droplet trajectory calculations, made prior to choosing the blade width, indicated that droplets would have to be quite small to pass through the blade passages without impact. As the blade width increases, the radius of curvature of the blade surface is increased. Such increases enhance the probability that a given size droplet will pass through the blade passage.

Considerable lap, i.e. blade height in excess of the nozzle height, was prescribed to capture the flow, particularly the droplets exiting the corner of the nozzle uppermost and in the direction of blade motion, since the droplet trajectories would be approximately straight (in the absolute frame) leaving the nozzle. This feature also helped to mitigate the effects of the large divergence of the nozzle in the radial direction, Figure 6. The shroud and shroud extension assisted in a rather abrupt straightening of a portion of the flow. Straight nozzle wall exit divergence was used because no two-dimensional calculational tools were available to reliably predict nozzle contours to provide parallel exit flow. All previous experiments had employed straight conical exits.

Blade spacing is typically smaller for a partial admission turbine than for a full admission turbine design. Blade spacing was chosen to be smaller than would be used in a standard full admission vapor turbine design [22]. This was done to improve partial admission performance and to improve the guidance of the vapor. It may be argued that increasing the blade spacing would allow more droplets to pass through the blade channel without impact. If the vapor were not coupled to the droplets, this argument would be uncomplicated, but since the vapor is affected by the droplets, their migration causes the vapor to migrate through momentum coupling. This results in not only a complex three-dimensional flow field but also in questionable net gains with respect to improvement of droplet kinetic energy conversion efficiency as a function of increased blade spacing.

Symmetric blades with nominal constant width channels were used. No attempt was made to design for a small positive reaction in order to improve pressure gradients for boundary layer considerations. The behavior of the two-phase boundary layer, the roughened liquid layer substrate with probable liquid entrainment due to droplet spallation and evaporation and rivulet formation due to coriolis and centripetal acceleration was not analytically tractable. It was anticipated that small negative reaction and unfavorable pressure gradients resulting in possible separation of the vapor boundary layer would be present. These effects were expected to be small compared to the potential losses associated with the liquid phase.

The blade contours were made with simple cylindrical surfaces for ease

of manufacturing. It was reasoned that non-uniform radii of curvature in the blade channel would detract from droplet performance. For a given blade turning angle, a non-uniform blade passage curvature requires a radius of curvature, which is at some point along the flow path, less than that of a simple cylinder. An increased probability that the droplet will strike the pressure surface results. No radial correction was made for changes as a result of either radial pressure gradients or local blade velocity in the relative velocities of the liquid and vapor.

Based on estimated slip ratios in nozzles [23], an approximate slip ratio of 0.8 was assumed. In addition to the estimated slip ratio, it was also assumed that the vapor relative velocity would be aligned with the blade pressure surface under optimum conditions. From this, the relative entrance velocity of the droplets could be estimated. A minor contour change was made to the blade profile in order to reduce the probability of droplet/blade impact in the entrance region of the suction surface, Figure 8. A slight angle modification was made in the straight portion of the suction surface contour to minimize the blade section in the assumed relative droplet entrance direction. For simplicity, this contour was repeated in the exit region also. The geometrical definition of the blades and rotor is summarized in Table 3 and Figure 8 and 9.

Fig. 8. Modification of simple suction surface for assumed droplet direction and blade details.

Table 3
Geometry details for figures 8 & 9

Blade suction surface radius, r_s	0.0444 m
Blade pressure surface radius, r_p	0.0571 m
Blade tip radius, r_t	0.00025 m
Blade height, H	0.1219 m
Blade width, B	0.1016 m
Overall rotor radius, r_o	1.034 m

Fig. 9. Rotor geometry.

Other Features of the Machine

The rotor and case assembly are shown in Figure 10. The rotor was designed to be supported by four sets of oil mist lubricated angular contact ball bearings. The bearings and lubrication system were chosen to minimize friction and because they were proven manufactured items. The bearings were supported with Bellville washers such that axial displacements of plus or minus 1 mm could be tolerated. The assembly was referenced to the duplex bearing which also provided thrust support. The large spherical boss (Figure 10) was used for final alignment of the rotor shaft in the assembly. Rotating face seals of the type used in the refrigeration industry were employed for the sub-atmospheric operation. The surrounding steel case was a bolted assembly in order to allow access to the rotor.

The total lower (or exit) side of the blade region was left open to the fluid collection volume in the rotor case. Though it could be argued that closing the inactive portion of this region could reduce disk friction and windages losses, it was thought that liquid buildup on such a closure might provide a mechanism for a larger loss (and possible damage) if entrainment into the rotating blades occurred. A simple turning guide and return channel were provided for collection of the vapor below the blades and approximately opposite the nozzle. No attempt was made to diffuse the vapor exiting the rotor.

The nozzle for the apparatus was suspended through the use of a four-bar linkage so that axial movement could occur. Fluid was

Fig. 10. Structural details of expander.

Fig. 11. Nozzle suspension for thrust measurement during expander testing.

supplied through a flexible pipe shown in Figure 11. A load cell was used to measure the axial thrust produced by the nozzle during performance testing of the machine.

Selection of Fabrication Techniques and Materials

Because of the relatively short lifetime requirements and the purpose of the machine, it was decided that an attachable blade design was an unnecessary complication for the laboratory. It not only represented additional design time, but also additional fabrication time. If the hardware was purchased outside of the laboratory, scheduling and associated costs appeared unreasonable with respect to the management constraints.

Based upon these considerations, an integral hub/blade design emerged. It was proposed that the most expedient way to fabricate the rotor was by machining it within the laboratory using a 5-axis numerically controlled milling machine. Structural considerations (to be presented later) required that the material have a high strength-to-weight ratio. Since heat-treatment for attainment of improved structural properties often causes local material deformation, the material also had to be heat treated prior to machining, and machinable in the heat treated condition. Operating temperature was not a design constraint because of the low operating temperature of the rotor (Table 1).

The choices of materials having high strength-to-weight ratios include:

high strength steels and various alloys of titanium and aluminum. High strength steels can be removed from the list because of poor machinability. Titanium alloys are more difficult to machine than those of aluminum and would have significantly limited the geometry and perhaps the choice of machining techniques. Aluminum, on the other hand, exhibited poor erosion resistance. Anodizing provided very little protection, even under the short lifetime requirements. Investigations [24] of various coating materials lead to the choice of electroless nickel as a coating material for aluminum. In order to maximize the strength-to-weight ratio, 7075 T-6 aluminum was chosen.

Major Aspects of Fabrication of the Rotor

The rotor was made from a 1,350 kg. forged aluminum disk. It was roughed to shape, ultrasonically inspected for flaws, stress relieved, and then heat treated and aged to the T-6 condition, prior to machining the blade profiles. Blade profiles were machined with a 1.27 cm. diameter milling cutter, as shown in Figure 12, to full depth. Figure 13 shows a view of the complete milling machine during the rotor machining operation.

Even with the stress relief, significant residual stresses were in evidence during the blade milling operation. The blades all bent forward (relative to the intended rotation direction) approximately 0.4 mm. The uniformity of the deflection from blade to blade was presumably the result of the symmetry of the forging. Because finish

Fig. 12. 1.27cm diameter cutter machining 12.20cm deep blades.

Fig. 13. Numerically controlled milling machine during rotor machining operation.

machining had not taken place, it was possible to relocate the effective center of the rotor and to complete finish machining. All blades were thus tilted forward slightly with respect to their originally intended positions.

Though the blades were finish machined, the final disk contours were not completed at this time in order to provide additional attachment locations for the electroless nickel plating process. A dye penetrant inspection established that there were no perceptible flaws or cracks in the blade root region and disk fillet regions.

A 0.075 mm nickel plating was applied to the rotor and blades [25]. The process was not only complex and time consuming, but required eight large tanks (Figure 14), an overhead crane, approximately six skilled personnel and in the final analysis, about six months. The actual plating process required approximately 12 hours with additional 12 hours preparation. A large fraction of the six months was used in attempts to have the plating done by outside firms. It was not possible to find a shop which had sufficient control or interest in the "one of a kind" job required to guarantee the results with specified certainty. For this reason, requisite modifications were made within LLL shops to accomodate the effort.

Having completed finish machining of the rotor, the finished steel hub/shaft assemblies were shrunk and bolted into place. The "as assembled" total shaft runout from end to end (approximately 75 cm.) was less than .04 mm.

Fig. 14. Plating line overview for application of electroless nickel.

Fig. 15. Overview of balancing operation.

Fig. 16. Screws being inserted in blade tips to provide balance corrections.

Fig. 17. Rotor assembled in case.

Fig. 18. Assembled expander showing nozzle with front plate of nozzle box cover removed.

Balancing of the completed rotor assembly was accomplished at Westinghouse in Emeryville (Figure 15). Weight corrections were made using lead wire and double locking set screws in threaded holes provided in the end of each blade (Figure 16).

The machine case and rotor were then assembled, aligned, and prepared for initial spin testing. The completed assembly without the front half of the rotor housing is shown in Figure 17. The final assembly, with the nozzle installed is shown in Figure 18.

Structural Analysis

Initial rotor design was based upon higher blade speeds than those finally required. Hence, the hub geometry evolved to that shown in Figure 10. A central hole was not used because of the initially high rotor design speed, and because hole growth, as a result of body forces during spinning, was large enough to cause unsatisfactory alignment for a simple shaft attachment. The hub attachment shown provided good alignment at all speeds, and reduced the central hub stresses in the aluminum compared to a disk with a center hole. A shrink fit of approximately 0.1 mm was imposed at the steel/aluminum cylindrical interface to assure concentricity. Design calculations were finally based upon a rotational speed of 6,000 rpm which is considerably above the maximum expected operating speed for the single nozzle laboratory design operating conditions.

Fig. 19. Simulation of blade loading for the rotor.
(displacements multiplied by 50)

A two-dimensional finite element structural analysis code, DTVIS2 [26] was used to analyze the stresses and displacements in the aluminum rotor and steel hubs. The code, developed for viscoelastic as well as elastic materials can be used to calculate spinning bodies of revolution and can include time- and temperature-dependent behavior.

Modeling of the blade loading on the disk (also see Appendix 3) required that an axially non-uniform radial load be imposed at a location corresponding to the base of the blades such that the outer rim at this location would remain parallel to the axis. This was necessary to properly account for the blades in the two-dimensional configuration, see Appendix 2. The average radial loading which would be imposed by the centripetal forces associated with the blades was applied at the outer rim (Figure 19) while the centripetal body forces within the remainder of the rotor were accounted for by using the spin option in DTVIS2.

Calculations were made at two operating temperatures. The 54 degree C case modeled operating conditions while the 100 degree C case examined accidental loss of vacuum during operation. The results of the stress calculations are summarized in Tables 4, 5, and 6, and in Figure 20. The high stress region for the aluminum is in the area marked 3 of Figure 20. This region can be easily inspected for flaws and surface finish. For this reason it represents an improvement over a simple solid disk, in which maximum stresses occur at the center. Material properties and overall displacements are shown in Tables 7 and 8, respectively. In addition to these calculations, transient thermal

Table 4
Stress in maximum stress regions of
Wheel and Hub @ 16°C and 6000 RPM

Region material	Radial MPa	Axial MPa	Hoop MPa	Max. prin. MPa	Von mises MPa
1 Steel	138	30	83	138	69
2* Steel	-11	90	165	83	172
2* Aluminum	-12	19	47	44	42
3 Aluminum	117	43	90	124	110
4 Aluminum	69	-28	69	69	96

Table 5
Stresses in maximum stress regions of
Wheel and Hub @ 54°C and 6000 RPM

Region material	Radial MPa	Axial MPa	Hoop MPa	Max. prin. MPa	Von mises MPa
1 Steel	221	23	110	227	110
2* Steel	-15	159	262	110	290
2* Aluminum	19	165	66	165	63
3 Aluminum	138	76	103	124	110
4 Aluminum	69	-41	66	62	110

Table 6
Stresses in maximum stress regions of
Wheel and Hub @ 100°C and 6000 RPM

Region material	Radial MPa	Axial MPa	Hoop MPa	Max. prin. MPa	Von mises MPa
1 Steel	283	43	124	296	331
2* Steel	-28	290	427	330	448
2* Aluminum	48	262	124	276	103
3 Aluminum	138	117	96	138	103
4 Aluminum	69	-69	69	69	124

- High stress in this region is thought to be an artifact of the zoning and the finite element modeling which does not allow sliding between the steel and the aluminum.

Fig. 20. Critical stress regions for hub and rotor.

Table 7
Material properties for Hub, wheel, and blades

	Elongation percent	Yield MPa	Ultimate MPa	Blade root design endurance limit MPa
Aluminum (7075 T-6)*	6.8	379	483	71.7
Steel (420 stainless)	8 (Rockwell C. Hardness = 48)	1,241	1,655	--

Table 8
Overall displacements

	@ 100°C-6000 RPM mm	@ 16°C-6000 RPM mm
Maximum axial displacement	0.21	-0.12
Maximum radial displacement	1.65	0.41

Fig. 21. Critical stress regions for blade attachment.

stresses were investigated to determine whether more severe stress requirements in the region of the hub could be imposed during the process of heating to operating temperature. Maximum transient thermal stresses were well below the yield strengths of both materials.

The steady radial blade attachment stresses were calculated assuming centripetal loadings on the blades modified with appropriate stress concentration factors to account for fillet radii in the region of the blade/rim interface. The forward deflection of the blades resulting from residual stress relief during machining tends to reduce the stresses in the critical blade root region, Figure 21, near the blade leading and trailing edges. As the centripetal force on the blade becomes higher, it tends to place the critical region in compression.

The dynamic bending portion of the critical blade stress near its root (Figure 21) was calculated based upon the conservative approximations of Kroon [27]. In this work, the stresses in the blade due to steady state application of the nozzle flow are magnified by an amplification factor based upon the worst possible loading condition and the damping properties of the structure (see Appendix 3). The partial admission applied loads are approximated as square pulses and are spaced at integral halves of the natural period of blade vibration. For the actual design, the loading would not be a square pulse, neither would it consistently occur at even half-periods. In addition, any water being radially removed from the blades in the inactive portion of the rotor will both change the blade natural frequency and tend to improve damping. Though Kroon conservatively used inherent material damping in

his analysis, structural damping properties were used herein. Measurements of the structural damping at low amplitude (which should be conservative), were initially accomplished using machining practice parts made prior to fabrication of the rotor and later verified using the rotor (Appendix 3). Nominal bending stresses near the blade root were estimated using beam theory, and modified with appropriate stress concentration factors. As an additional check, strain gages were installed in critical areas and static loading was applied to corroborate the calculated stresses.

In order to assess the ability of the blade root region to withstand the fatigue conditions associated with the oscillatory loading, it was necessary to combine the effects of the steady and oscillatory loading using the Soderberg law. Appropriate design factors were estimated [28] and applied to the raw fatigue data for the specific alloy, heat treatment, and geometrical configuration. The stress resulting from the combined loads is in the safe stress region for all operating conditions of interest (Appendix 3). A summary of the stresses in the critical blade root region is presented in Table 9.

Critical structural natural frequencies and exciting frequencies are shown in Table 10 and Figure 22. The blade fundamental frequency was estimated to within less than 5 percent of the measured value using Rayleigh's method. Other frequencies were estimated using the appropriate approximations and were later verified experimentally. The nozzle passing frequency is near the blade fundamental frequency at

Table 9
Maximum blade root stresses at
turbine speeds less than
6000 RPM

	Stress—MPa
Centripetal stress	114
Oscillatory stress	15.6

Table 10
Estimated natural frequencies
for the structure

	First mode Hz	Second mode Hz
Disk	360	2000
Blade	1400*	5000*

• Measured

Fig. 22. Nozzle passing frequency versus rotor speed.

approximately 2,700 rpm. The rotor fundamental rotational frequency excites the fundamental frequency of the machine mounting at 2,000 rpm.

Static Cascade Design Considerations and Description

As previously discussed, management placed significant emphasis upon building the dynamic blade test apparatus. During the time fabrication was proceeding, limited static cascade experiments were made using the same blade contours as would be used in the dynamic blade test.

Static cascade experiments were limited for several reasons. The flow available in the test facility where the cascade experiments would be executed was limited, leading to a cascade consisting of only a few blades. Nozzle design considerations dictated that if the velocity of the two-phase mixture relative to the blades was to be simulated, large droplets would be likely to occur. Unrealistically large droplets, even at the correct velocity relative to the blade, would not properly simulate the conditions encountered in the rotor. Increasing the enthalpy drop across the nozzle to simulate actual operating conditions and to provide realistic droplet size meant that the relative vapor velocity would be high by a factor of approximately two. Small droplets at this higher velocity also provide a poor simulation of the conditions found in the rotor. In addition, relative velocity (direction) differences (Figure 5a) between the vapor and the droplets which would appear in the rotor cannot be simulated with a static cascade.

Fig. 23. Static cascade apparatus mounted for testing.

The "cascade" consisted of three blades mounted so that a low velocity two-phase nozzle [29] exhaust could be directed through the blade passages. The flow conditions for the static cascade tests were selected to simulate the relative velocity and quality expected in a full admission turbine operating near peak efficiency. Boiler stability required that the exit pressure for the nozzle be higher than would be used for the dynamic blade test. The estimated exit conditions were slightly subsonic relative to the vapor. The blades were supported, Figure 23, in a four-bar mechanism such that blade thrust could be measured. The blades were also fitted with small static pressure taps for the purpose of measuring local pressures within the blade channel.

Numerical Performance Modeling

As noted previously, the most comprehensive approach to understanding two-phase fluid dynamics is through a completely coupled solution. In the coupled solution, there are two levels of sophistication which are summarized in Table 11 for two-dimensions. The most difficult case assumes that thermal equilibrium between phases does not exist which leads to ten equations for characterization of the system. In the thermal equilibrium case, the two phases are assumed to be in temperature equilibrium, implying saturation properties— thus simplifying the solution to only eight equations. In both of the above cases, at least four of the equations are partial differential equations relating continuity, momentum, and energy. These equations

Table 11
Coupled two-phase, two-dimensional
equations and variables
for single-component flow

Temperature nonequilibrium between phases	
10 equations:	2 continuity 4 momentum 2 energy 1 heat transfer equation 1 equation of state
10 variables:	Pressure Mixture density Mass fractions of vapor Vapor velocity x-direction Vapor velocity y-direction Droplet velocity x-direction Droplet velocity y-direction Temperature of vapor Temperature of droplet Droplet diameter
Temperature equilibrium between phases	
8 equations:	2 continuity 4 momentum 1 energy 1 equation of state
8 variables:	Pressure Mixture density Mass fraction of vapor Vapor velocity x-direction Vapor velocity y-direction Droplet velocity x-direction Droplet velocity y-direction Droplet diameter

may be written for the averaged droplet phase, the vapor phase, or the sum of the droplet and vapor phase or mixture equation. The partial differential equations are written in the Eulerian frame. The droplet phase can also be represented in the Lagrangian frame by ordinary differential equations [30] and, thus a choice of equations exists for the complete solution, as indicated in Table 12.

The complete coupled solution has been accomplished in two-dimensions for shockless flows [12] and in one-dimension for flows including shocks [7]. A two-dimensional coupled model, specifically designed for two-phase blade and nozzle calculations, is in progress. A two-phase boundary layer solution for droplet in vapor flows is being developed [31] but is not presently available.

Coupled models are not only difficult to develop because of their complexity, but they require large amounts of computer time. In addition, the approximations of the detailed physics which must be used, particularly in the case of droplet/wall interaction, make even a coupled representation somewhat dubious unless it is assisted with detailed experimental support.

In order to examine gross machine performance, other effects, in addition to flow through the blade channel, must be addressed. The loss mechanisms which should be represented in a performance model are listed in Table 13. The first three shown are associated with any turbomachine one might design. The latter three, however, are specific

Table 12
Possible combinations of equations for
coupled two-phase single component flow

	Eulerian vapor	Eulerian droplet	Lagrangian droplet
Eulerian mixture	•	•	•
Eulerian vapor		•	•

to the two-phase design under consideration.

Table 13
Turbine loss mechanisms

Boundary layer and disk friction
Turbulence and friction losses in nozzles and blades
Shock losses
Droplet/wall interaction losses
Liquid pumping losses
Slip (relative velocity between liquid and vapor phase)

It can be observed that all three of the loss mechanisms associated with two-phase turbodesign are specifically related to the behavior of the liquid phase. It is imperative to understand the behavior of the liquid as it passes through the machine since its presence is the most fundamental complicating factor in the design and performance prediction. The performance model to be presented addresses the behavior of the liquid and also that of the vapor.

A model for blade flow which approximates the coupled solution but which requires less computation time must be a part of the performance model desired. It has been found that the local vapor velocity profile in the blade passage is of secondary importance to the droplet trajectory (see Appendix 5) for all conditions of interest herein. Capitalizing upon this observation, one may assume that the vapor

velocity between blades is uniform and constant for the case of the impulse turbine. Droplet trajectories can then be calculated on the basis that the vapor does effect the droplet through drag but that the droplet does not effect the local state or velocity profile of the vapor. By calculating several trajectories, the two-dimensional behavior of the liquid droplets can be approximated. This allows one to determine whether or not droplet impact with the blade occurs, and thus what fraction of the liquid strikes the blade. Droplets which impact the blade are assumed to stick to the surface. This assumption is supported by static cascade results (to be presented later). Some small amount of liquid spallation is likely to occur; however no satisfactory way of estimating it exists; and it therefore is neglected. The static cascade experimental results also indicate that spallation is not large. Vaporization of liquid which results both from droplet impact with the blade surfaces and from vapor enthalpy gain due to friction is accounted for. Friction effects on the vapor flow through the blade channel were determined experimentally using static cascade results. As an alternative, the model also allows the use of loss coefficients based upon standard vapor turbines which implicitly assumes that the presence of the liquid does not alter the loss coefficients. Changes in pressure and droplet diameter are neglected through the blade channel. The pressure at the entrance and the exit to the blade channel is assumed to be the same. Finally, droplets which pass through the blades, without impacting their surfaces, are assumed to be small enough so that there is no exit slip.

Having found the dispersment of the liquid, it is possible to

approximate the correct exiting vapor velocity by using a two-phase mixture energy equation written for a control volume whose boundaries are inside the boundary layers of the blade channel (in the relative frame). This approach provides for the characterization of the liquid and vapor flow in the blades with emphasis on what happens to the droplets. The complex, fully coupled solution has thus been replaced by two Lagrangian droplet momentum equations and equations of mixture continuity and mixture energy. The detailed equations and solution technique for the simplified approximation described may be found in Appendix 5.

Oblique shocks which may be present within the blade channels are not accounted for. Because the calculations assume ideal vapor incidence only, incidence shock corrections are not required, and the sharp edged blade profiles of interest here should minimize shock losses. The importance of accounting for two-phase shock phenomena will increase with decreasing droplet size because as the droplets become smaller the mixture will approach the behavior of a dense gas (see Appendix 6).

Velocity vector equations for each phase are used to represent the entrance and exit conditions for the rotor. The inlet is determined by the nozzle while the exit conditions are modified by the blade flow approximations. For the purposes of calculation, it is assumed that the blade geometry is fixed but that the nozzle angle is adjusted for the speed of the rotor so that the vapor always enters the blade channel in a direction which is tangent to the blade pressure surface. Thus, a curve of engine efficiency versus rotor speed calculated in

this manner would not be representative of a machine of fixed geometry design because of fixed nozzles and because of incidence loss. However, the peak of such a curve is consistent with a properly designed experimental geometry. The effects of radial pressure gradients and the radial variation of the velocity vector triangles are not included. For large droplets, rotor induced radial pressure gradients should be related to the vapor density because coupling between phases is poor; for small droplets the pressure gradient should be related to the mixture density. In either case, the change in enthalpy as a function of pressure in the two-phase mixture region is considerably less than in a pure vapor region. Thus, the radial reaction variation is less severe in the present regime than would be expected if the working fluid were vapor only. It is assumed that the droplets entering the rotor are all the same size and that the flow is one-dimensional.

In order to put the blade flow calculations into the context of the complete machine performance, additional assumptions and relations must be employed. Nozzle flow must be modeled in order to provide information concerning flow conditions entering the turbine rotor. The fluid entering and exiting the rotor must be viewed from the absolute and relative frames in order to establish gross output power. The loss mechanisms (see Appendix 5) previously mentioned must also be accounted for so that expected performance can also be predicted.

Two methods were used to characterize the nozzle. In the first method, 1D2FAZ was used to determine the ideal core flow (not including

boundary layers). The Weber number break-up criteria was adjusted so that the calculated thrust coefficients agreed with those determined experimentally. Thrust coefficient is defined in Appendix 4, (4.2). This calculation thus produced a representative droplet size. The Weber number criteria is not adequate for single component flows, but it was the best approximation available at the time of the calculation. A second technique (see Appendix 4) of nozzle characterization consisted of using the exit area, the measured mass flow-rate, the measured thrust coefficient, and the measured exit pressure to solve the two-phase mixture equations of continuity momentum, and energy at the entrance and exit plane of the nozzle. The second technique (referred to as "nonequilibrium") was generated when it was observed that the mass flow rate calculated by 1D2FAZ was 10 to 20 percent less than that measured experimentally. It was postulated that the increased mass flow rate was due to nonequilibrium at the throat of the nozzle. Thus, in the throat, vapor is produced by evaporation more rapidly in the 1D2FAZ calculation (which presently assumes thermal equilibrium between phases) than in reality, causing choking to occur at a lower flow rate. The actual conditions in the throat appear to be considerably out of thermal equilibrium, based on the ratio of nozzle throat transit time to the heat transfer time constant for droplets of the diameter estimated. On the other hand, comparison of time constants for droplet heat transfer and for total nozzle transit indicate that it is reasonable to expect thermal equilibrium at the exit of the nozzle—which is the basis of the nonequilibrium technique. A compressible boundary layer was calculated using [32] which is based upon the pressure profile and average one-dimensional two-phase mixture

density from the 1D2FAZ calculation. Boundary layers were found to be responsible for less than a one percent momentum loss, and were thus neglected.

Having modeled the blade passage and nozzle flows, all that remains is to combine these with the overall expander performance. To do this, the two-phase mixture continuity and turbine energy equations are written in the relative and absolute reference frames for the entrance and exit conditions of the rotor to determine gross power output. The gross power is corrected to engine efficiency by subtracting the power associated with disk friction and windage and liquid pumping loss resulting from radial removal of droplets adhering to the blade surfaces. For flows of large droplets, coupling will be poor which will lead to removal of droplets from the mixture. For this case (particularly for partial admission) vapor density should control disk friction and windage behavior. In the case of small droplets, the additional coupling may cause the mixture density to be in control. Blade tip clearance and sector losses are also accounted for assuming that they affect the vapor phase only. The details of all the above considerations are presented in Appendices 4 and 5.

Discussion of the Model

The model presented is specialized. Its intended emphasis is

specifically to account for the liquid phase--though the vapor phase must be and is included. A fully coupled model would require significantly more computation and development time.

A single droplet diameter is used in this presentation to represent a more complex droplet distribution. This may seem simplistic. The droplet distribution could be used by the performance model, if it was known. Presently, neither the spectrum nor even a representative droplet diameter has been experimentally measured. Calculational determinations of the kind described previously are, at best, estimates. Accepting the approximate nature of droplet sizes, a great deal can be learned when one uses a representative droplet size as a means of establishing bounds. Thus a representative droplet diameter can be used to establish limiting performance characteristics, as will be shown later.

The liquid behavior is treated conservatively, particularly droplet impact. In addition to possible spalling of impacting droplets, splashing of thin rivulets of water might reduce the amount of water required to be removed radially from the blades. Estimates of velocities of a liquid film [33] in the direction of vapor flow indicate that even if it was driven throughout its length by impacting droplets, the liquid velocity would be less than 15 percent of that of the impacting droplets. This finding is consistent with a two-dimensional liquid film analysis, presented in Appendix 5, which indicates that the liquid film must be driven over the total blade pressure surface to escape being forced to exit radially. It is

expected that liquid will move in a predominately radial (along the length of the blade) direction, maximizing the loss associated with radial liquid removal (Coriolis pumping loss).

Treating the vapor portion of the flow with vapor turbine blade loss coefficients would be optimistic since a liquid structure of any kind on the blade surface would contribute a rough surface as well as possible mass injection resulting from local instabilities. Fortunately, it was possible to correlate vapor friction loss coefficients to static cascade experimental results through use of this performance model.

The performance model is ultimately intended to be combined with a numerical optimization routine [34] in order to predict the geometry for a Total Flow turbine of maximum efficiency. When droplet spectra are experimentally characterized, coupled two-phase blade calculations, two-phase boundary layer calculations, and additional experimental two-phase flow measurements are available for "calibration" of the performance model, numerical optimization will become quite useful.

Static Cascade Testing And Analysis

The static cascade experimental results have contributed significantly to the understanding of two-phase turbine performance. Because of the limitations described previously, the static cascade testing cannot replace the dynamic blade apparatus testing. However, as will be seen, much inferential information can be obtained from the static cascade measurements.

Thrust measurements were made with blades which were wider than the nozzle to simulate lap, with spacing at one end of the blades to simulate rotor clearance. Pressure measurements were made on blades which matched the nozzle height and were sealed at both the top and the bottom of the blade channel. A strain-gage type transducer mounted in a Scannivalve was used to make pressure measurements.

The results of thrust measurements are plotted in Figure 24. Relatively large changes in incidence angle did not significantly affect the thrust performance of the cascade. Also, doubling the blade spacing did not significantly change the results.

The measured pressure profiles are shown in Figure 25. Near the entrance to the blade channel, on the pressure surface, there is a marked increase in pressure. This is attributed to droplet impact, indicating that most of the droplets impacted the blades in this experiment. The slight positive pressure on the suction surface is

Fig. 24. Measured static cascade thrust for various conditions.

Fig. 25. Measured pressure profiles from static cascade.

estimated [35] to be the result of cross-flow momentum coupling of the droplets with the vapor.

In order to better understand the results, the blade flow portion of the performance model was employed. Nozzle performance was first characterized as described earlier, in order to set the initial conditions for the blade flow estimates. The results of the nozzle characterization are listed in Table 14 for both the 1D2FAZ calculation, and the nonequilibrium calculation. Droplet diameter, based upon matching the measured thrust coefficient with a Weber number of 16 in the 1D2FAZ calculation, was predicted to be about 9 times larger than would pass through the blade passage without impact. This is consistent with the blade thrust measurements which indicated the same thrust coefficient for the nominal blade spacing as for double the nominal spacing.

Simulations of cascade thrust were made with the performance model for both nozzle performance estimates. In order to match the experimental results, it was necessary to assume that all droplets struck the blades and that their velocities were reduced to negligible values, which was in keeping with the droplet size estimate. Experimentally derived values of the vapor relative velocity coefficients were determined in this fashion from the model. It was also assumed that vapor friction increased the temperature of the vapor at the exit of the rotor (resulting in a slight vapor superheat), rather than causing additional water evaporation. Comparison of the vapor relative velocity coefficients for the two nozzle characterizations, Table 15, indicates

Table 14
Cascade nozzle characterization

Inlet conditions	1D2FAZ calculation	Nonequilibrium calculation
Pressure MPa	0.503*	0.503*
Mass flow rate ratio	0.28	0.28
Vapor velocity m/s	29	34
Slip ratio	1	1
Mass flow rate kg/s	0.410	0.471*
Exit conditions		
Pressure MPa	0.100	0.100*
Mass flow rate ratio kg/s	0.330	0.331
Vapor velocity m/s	426.7	485.1
Slip ratio	0.780	0.616
Droplet diameter μm ($We = 16$)	19.5	—
Thrust coefficient (measured thrust/ideal isentropic homogeneous thrust)	0.87	0.86*

* Measured

a consistent value of approximately 0.75. For a pure vapor case, impulse blading correlations [36], [22] indicate that the coefficient would be 0.87 to 0.93.

Table 15
Vapor velocity coefficient correlations
assuming droplets all impact blades

λ_v = Ratio of vapor velocity at the entrance divided by that at the exit	
Nozzle characterization	λ_v
1D2FAZ	0.745
Nonequilibrium	0.770

Pressure profile measurements were also analyzed with the performance model. If it is assumed that the difference in pressure between the suction and pressure surfaces (Figure 25) has two components: one consisting of the centripetal acceleration of the vapor, and the other resulting from the impact of droplets of various sizes, some insight into droplet size and flow distribution is possible. Ideal average liquid phase impact pressures for three different droplet sizes are shown in Figure 26 compared to the measured pressure difference. These profiles were calculated based upon the 1D2FAZ nozzle estimates, using droplet trajectory information within the performance model (see Appendix 5). The pressure difference resulting from the centripetal

Fig. 26. Comparison of pressure difference between the pressure and suction surfaces and the calculated average droplet impact pressures.

acceleration of the vapor (at the blade exit vapor velocity) is also shown.

The comparison of experimental and calculated results in Figure 26 shows that the flow exiting the nozzle contains droplets which may be between 2 and 20 microns. The accentuated peak indicates possible nonuniformity of the liquid distribution in the nozzle. This would be expected, based on experimental results obtained in gas/solid flows in nozzles [37]. The nonuniform liquid distribution is further corroborated by noting that the integral of the difference in pressure force in the direction of "blade motion" is considerably less than would be indicated by the thrust measurement. In order to match the thrust measurement the pressure difference had to be significantly greater at locations other than those measured. Only one blade channel was instrumented and the pressure taps were located in only one plane. It is thus reasonable to believe that nonuniformity of the liquid distribution, based upon the lack of correlation of the thrust with the measured pressure difference force, could have been severe.

Dynamic Blade Apparatus Testing and Analysis

The dynamic blade test apparatus was attached to a hydraulic pump/motor [38] through a torque sensing coupling (Appendix 7). The hydraulic system was operated as a brake for determination of power output and as a drive unit for measurement of disk friction and windage. The drive unit mode was powered by a diesel driven pump.

Fig. 27. Control room and data collection equipment.

In addition to the torque sensor, a digital revolution counter was used to determine angular velocity. Numerous displacement transducers, accelerometers, thermocouples, and pressure transducers were also used to measure data which was ultimately collected in the control room (Figure 27) and recorded on both paper and magnetic tape. A summary of the instrumentation and its purpose may be found in Appendix 7.

Disk friction and windage experiments were made in both air and water vapor at a variety of pressures. Testing was done at several pressures in order to broaden the range of Reynolds numbers over which the tests were run. The moment coefficients derived from the data are plotted as a function of Reynolds number in Figures 28 and 29. The Reynolds number and moment coefficient for the rotor are defined in Appendix 5, (5.65) and (5.67), respectively. The temperature in the rotor chamber was monitored during the air tests and the results were used to correct for density and viscosity. It will be noted that the data for air and the data for water vapor results in two separate curves. It would be expected that if the both fluids behaved as perfect gases during the tests, that only one curve would emerge. Because the water vapor curve lies above the air curve, it is hypothesized that condensation occurred in the blade region, due to reduced local pressure between blades. The occurrence of condensation provides two mechanisms: condensed liquid which must be radially removed increases the moment coefficient, and reduced density in the region of condensation which could induce local laminar behavior by lowering the local Reynolds number. The latter mechanism explains the increased slope of the water vapor curve at lower Reynolds number.

Fig. 28. Moment coefficient versus Reynolds number for rotor in air.

Fig. 29. Moment coefficient versus Reynolds number for rotor in water vapor.

Power measurements were made on three separate occasions and showed good repeatability. Operation of the apparatus was quiet and smooth. The measured data are plotted in terms of engine efficiency versus blade speed ratio in Figure 30. Engine efficiency is defined as the turbine specific power output divided by the specific power associated with an isentropic expansion between the nozzle inlet state and the turbine exit pressure. The blade speed ratio is defined as the blade velocity divided by the velocity resulting from an isentropic expansion between the nozzle inlet state and the turbine exit pressure. Each data point is the average result of at least one minute of running at constant speed. Very little variation of speed and torque was observed during the collection of data for each point.

The measured thrust coefficients for the turbine nozzle under three operating conditions are listed in Table 16. It will be noted that the test conditions differ slightly from the original design conditions (Table 1). The final test conditions were necessitated by limitations of the two-phase flow test facility. The shroud and shroud extension (See Figures 31, 6, and Figure 4.2 of Appendix 4) caused some reduction in the thrust coefficient. It was observed [39] during nozzle testing that the flow pattern was sensitive to exit pressure. The exhaust flow pattern was observed to concentrate on one side of the nozzle or the other when off-design conditions were imposed by too high an exit plane pressure.

Nozzle thrust coefficients were first calculated using 1D2FAZ and the nonequilibrium model, as described previously, to predict the

Fig. 30. Measured engine efficiency as a function of blade speed ratio.

Table 16
**Measured performance for the dynamic
blade test nozzle**

Test configuration:	Nozzle alone (static test)	Nozzle with shroud and extension (static test)	Nozzle with shroud and extension during dynamic blade test
Inlet conditions			
Pressure MPa	2.426	2.447	2.528
Mass flow rate ratio	0.148	0.140	0.141
Mass flow rate Kg/s	0.580	0.585	0.596
Exit conditions			
Pressure MPa	0.0139	0.0142	0.0132
Thrust coefficient (measured thrust/ideal isentropic homogeneous thrust)	0.94	0.91	0.83
Pressure ratio (exit pressure/ inlet pressure)	0.0057	0.0058	0.0052

Fig. 31. Dynamic blade test nozzle with bottom plate removed.
Shroud and shroud extension are indicated.

conditions at the exit of the unshrouded nozzle, Table 17. The droplet size was estimated with 1D2FAZ as indicated previously, by adjusting the Weber number so that the calculated thrust coefficient agreed with that observed experimentally. This resulted in a Weber number of 8. It should be noted that the nozzle exit conditions for this calculation were found to be independent of the initial droplet size assumed at the entrance of the nozzle. The exit conditions (including droplet size) for calculations based upon 50 micron and 250 micron initial droplet diameters, other initial conditions being equal.

The reduction in thrust coefficient observed as a result of the shroud and shroud extension may be explained by hypothesizing oblique shocks resulting from straightening of the diverging flow at the exit of the nozzle, Figure 31. To estimate the validity of this hypothesis, a two-phase oblique shock solution was made (Appendix 4), based upon the 11.6 degree divergence (neglecting the 2 degree divergence in the perpendicular plane). This consisted of solving the appropriate two-phase equations of continuity, momentum, and energy with the equation of state for water, assuming that the droplets were large enough so that they did not change velocity through the shock. By making such a calculation for a variety of deflection angles equal to and less than the nozzle divergence, and momentum averaging the resultant velocities, one can determine the results as shown in Table 18. It appears that oblique shocks can well explain the effects of the shroud on the flow. An additional insight can be gained by observing that the calculation primarily involves a change in the state of the vapor. This indicates that the droplets are concentrated near the

Table 17
Dynamic blade test nozzle characterization

Case: Calculation:	1 1D2FAZ	2 Nonequilibrium	3 1D2FAZ oblique shock	4 Nonequilibrium oblique shock
Configuration:	Nozzle alone	Nozzle alone	Nozzle, shroud and shroud extension	Nozzle, shroud and shroud extension
Entrance conditions				
Pressure MPa	2.528	2.528*	2.528	2.528*
Mass fraction	0.141	0.141	0.141	0.141
Vapor velocity m/s	14.2	15.5	14.2	15.5
Slip ratio	1	1	1	1
Mass flow rate kg/s	0.544	0.596*	0.544	0.596*
Exit conditions				
Pressure MPa	0.0142	0.0132*	0.0142	0.0132*
Mass fraction	0.331	0.332	0.36	0.37
Vapor velocity m/s	697	814	642	744
Slip ratio	0.906	0.710	0.981	0.777
Droplet diameter μm ($We = 8$)	6.5	—	—	—
Thrust coefficient (observed thrust/ isentropic homogeneous thrust)	0.94	0.94*	0.91	0.91*

* Measured

center of the nozzle as was indicated in the static blade tests. If this was not the case, the presence of the shroud would also result in increased numbers of droplet/shroud impacts. Droplet shroud impacts would be expected to significantly reduce the velocities of the impacting droplets—hence requiring that change in the liquid phase velocity would have to be included to explain the experimental results.

Returning to Table 16, the nozzle conditions corresponding to the dynamic blade test will now be addressed. The shroud remained in place during these tests. One observes that the thrust coefficient is significantly lower for the dynamic blade tests than for previous nozzle tests. The pressure ratio is the only notable change. The thrust coefficient of this nozzle was shown [39] to be insensitive to pressure ratio variations even larger than these. The variation of the thrust coefficient for the range of pressure ratios shown in Table 16 was found to be less than 1 percent when only the nozzle was tested. Pressure conditions at the exit plane of the nozzle were different during the dynamic blade test than during the test of the nozzle alone. The rotation of the blades causes the fluid between the blades to develop a small radial pressure gradient along the length of the blade. The spacing of the blade and nozzle was less than 10 mm so that the pressure gradient could have caused one side of the nozzle flow to separate. Separation could result in the unstable flow behavior observed when the exit pressure was too high, i.e. flow concentration on the inner radial side of the nozzle passage.

It is hypothesized that this behavior could lead to two conditions

which can be approximated independently. The first assumes that the diversion of the flow to one side of the nozzle forces droplet impact to occur on the adjacent nozzle wall, reducing the impacted water droplet velocity to a negligible value. The second assumes that oblique shock conditions resulting during flow diversion cause the vapor and liquid to exit the nozzle at the same velocity, i.e. with no slip. In the first condition, the flow is assumed to be represented by cases 3 and 4 of the nozzle characterization (Table 17). From the definition of the nozzle thrust coefficient, the mass flow rate of liquid impacting the walls is then determined so that the observed nozzle thrust coefficient is replicated (see Appendix 4). For the oblique shock, the equal exit velocities of both phases are calculated to duplicate the observed thrust coefficient.

With the nozzle characterization thus established, the performance of the dynamic blade test apparatus can be predicted, within the uncertainty of the exit condition of the nozzle and the disk friction and windage, using the numerical performance model which was previously described. The relative vapor velocity coefficients used were based upon the simulation of the cascade tests, presented previously. The estimated size of the droplets, using 1D2FAZ correlations, indicated that the droplets were three times as large as the maximum size calculated to pass through the blade passage without impact. The velocities, fraction of droplet impacting the blade surfaces, and mass flow rate ratio represented in the static cascade experiment modeled well the conditions encountered in the dynamic blade test. The major difference was that the calculated vapor velocity relative to the vapor

sonic velocity was higher in the dynamic blade test than in the cascade test—primarily because the test pressure was lower. While the Mach number relative to the vapor sonic velocity was slightly less than 1 for the static cascade test, the Mach number relative to the vapor sonic velocity was calculated to be between 1 and 1.3 for the dynamic blade test. Though the two-phase mixture Mach number would be greater, the large droplet diameters estimated would indicate (Appendix 6) that two-phase flow adjustments would not result in discontinuous fluid behavior. In addition, the large droplets are rapidly removed from the flow by blade surface impact, resulting in essentially pure vapor flow over a large portion of the blade passage. This indicates the deterministic nature of the Mach number relative to the vapor, in this case, and provides additional support for the use of static cascade velocity coefficients in the dynamic blade test performance modeling. Adjustment of the nozzle angle prior to testing, based on performance estimates, allowed the calculated vapor incidence angle of the flow to be less than 2 degrees at the measured peak operating point, further minimizing the probability of shock formation. The sharp edges of the blades were specifically designed to minimize shock losses associated with the leading edge. Aside from possible oblique shock formation, the static cascade vapor velocity coefficients well represented the physical conditions expected in the dynamic blade test.

The bounding calculational results are shown in Figure 32 with the nozzle exit characterization shown in Table 18. The upper and lower bands result from assuming pure vapor and mixture density respectively, for the disk friction and windage modeling. The bounds for each band

Fig. 32. Comparison of numerical performance model predictions with measured performance.

Table 18
Bounding nozzle characterizations for
dynamic blade test simulations

Case:	1	2
Calculations:	1D2FAZ oblique shock (upper bound)	Nonequilibrium oblique shock (lower bound)
Blade entrance conditions:		
Pressure MPa	0.0132	0.0132
Mass flow rate ratio* kg/s	0.331	0.331
Vapor velocity m/s	642	744
Slip ratio	0.981	0.777
Droplet diameter μm	6.5	6.5
Thrust coefficient	0.83	0.83
Mass flow rate kg/s	0.596	0.596
Mass flow rate of liquid slowed to negligible velocity in nozzle kg/s	0.051	0.055
Blade vapor velocity coefficient	0.75	0.77

- Though the oblique shock calculation assumed thermodynamic equilibrium between phases, leading to the mass flow rate ratio quoted in Table 18, this calculation was performed assuming that heat transfer could not occur so that the vapor exiting the nozzle was slightly superheated.

enclose the performance predicted by the "no slip" nozzle calculation. The upper bound (Table 18) of each band resulted from 1D2FAZ predictions modified to include an oblique shock and liquid capture. The lower bound of each band resulted from the nonequilibrium model prediction modified to include an oblique shock and liquid capture. In both cases the nozzle-captured-liquid exiting the nozzle was assumed to be reaccelerated by the rotor through inelastic collision.

It is noteworthy that the blade speed ratio for the peak of the upper band is higher than the measured peak, and that the blade speed ratio for the peak of the lower band is lower than the measured peak. The blade speed ratio for peak performance of a pure impulse turbine can be shown to be independent of the relative vapor velocity coefficient and likewise the liquid capture. The peak efficiency shifts to lower blade speed ratios primarily as a result of disk friction and windage and liquid pumping associated with removal of captured droplets from the blades. Thus, the model does appear to be indicating that the peak of the experimental data includes more disk friction and windage loss than vapor alone could explain. This is a plausible result, since the droplets being thrown from the blades would be expected to splash on the rotor case (which is separated from the blade tips by only 10 mm). Thus the effective density of the vapor at the outside of the rotor is increased and there also exists the potential for reacceleration of entrained liquid by the blade tips, further enhancing losses.

In summary, the nozzle modeling for both the cascade test and the dynamic blade test was done with 1D2FAZ and the nonequilibrium

calculation. Droplet sizes were estimated using 1D2FAZ and the measured thrust coefficient. The results of cascade simulations indicated consistent vapor velocity coefficients for both nozzle simulations. Droplet size appears to include a range of from 2 to 20 microns in diameter. Comparison of static cascade results with the numerical performance model indicate that all droplets impact the blade surfaces and are reduced to negligible velocities. Nonuniformity of liquid distribution across the nozzle was also indicated by the results. The disk friction and windage measurements indicated that water vapor can produce higher moment coefficients than pure air. This is attributed to local condensation phenomena which causes liquid formation on the blades as well as laminarization. Oblique shocks were seen to be the probable cause for the reduction performance of the dynamic blade test nozzle when the shroud was installed. Nonuniformity of the liquid distribution was also indicated in this nozzle. Lower nozzle thrust coefficients during the dynamic blade test were attributed to radial pressure gradient caused redistribution of the flow near the exit of the nozzle. The poorer performance was modeled as being caused by liquid capture on the walls of the nozzle. The performance prediction indicated good agreement with the measured results. Disk friction and windage were found to be slightly greater than would be expected if only vapor environment surrounded the rotor--indicating liquid entrainment in the blade tip region.

Extrapolations, Using the Numerical Performance Model

Having shown the validity of the performance model with respect to the experimental data, it is useful to further investigate selected design options. The performance model was designed to include variable admission and droplet size, as described previously. Full admission results were estimated, for the same rotor and blade geometry as tested, using the same model as was used previously for predicting the experimental results—except it was assumed that the nozzle shroud was removed and that the nozzle was separated sufficiently from the rotor so that the 0.94 thrust coefficient, which was achieved during nozzle testing, could be achieved in the full admission configuration. The results of a full admission extrapolation with droplets too large (droplet diameter, $D > 2.3$ microns) to pass through the blades without impact is shown in Figure 33. The upper bound (solid lines) results from using the 1D2FAZ nozzle prediction (Case 1, Table 19) without the shroud and assuming that the vapor density controls disk friction and windage. The lower bound (solid line) was calculated based upon the nonequilibrium nozzle performance estimate (Case 2, Table 19) without the shroud and assumes the two-phase mixture density controls the disk friction and windage. The bounds predictions may be extended upward (dashed lines) because of uncertainty regarding the liquid pumping losses. It is not certain, when one considers the forces on a liquid film or rivulet (Appendix 5), whether the full admission condition will cause some or all of the liquid to reach the exit side of the blade without significant radial change. With the single nozzle, as tested, a particular blade is in front of the nozzle for less than 4 percent of

Fig. 33. Extrapolations using numerical performance model.

the time. Consequently, for the single nozzle test, forces associated with the flow of the mixture through the blades, which can be estimated to be of the same order as the centripetal force, cannot be as important as the centripetal force which acts continuously. In the case of the full admission configuration, however the forces associated with the flow of the fluid mixture through the blades are essentially continuous. The relative magnitudes of the forces acting on the liquid and the uncertainty as to whether the liquid is a film or a rivulet make it unclear whether the liquid impacting the blade leaves radially, leading to the full pumping loss term and the lower solid curve; or whether all of the liquid moves across the blade without significant radius change, resulting in the dashed curves. Calculations, based upon a two-dimensional continuous liquid film analysis (Appendix 5) and large droplets indicate that the lower performance is most probable.

Table 19
Full admission nozzle characterization

Case:	1	2
Calculation:	1D2FAZ	Nonequilibrium
Blade entrances Conditions		
Pressure MPa	0.0132	0.0132
Mass flow rate ratio kg/s	0.331	0.332
Vapor velocity m/s	697	814
Slip ratio	0.906	0.710
Droplet diameter μm	6.5	6.5
Thrust coefficient	0.94	0.94
Mass flow rate kg/s	13.7	13.7
Blade vapor velocity coefficient	0.75	0.77

The dependence of performance on droplet the diameter was also calculated through use of this model. Droplets which will pass through the blades without impact must be less than 2.3 microns in diameter. It was assumed that droplets of this size or smaller will have very little slip, relative to the vapor in the exit plane of the nozzle, so that the slip ratio was set to unity while the 0.94 thrust coefficient was maintained. The mixture density is used for disk friction and windage estimates because it is assumed that droplets less than 2.3 microns will cause the fluid surrounding the rotor to behave as a homogeneous mixture of droplets and vapor.

The research objective of the Total Flow conversion development is the achievement of 70 percent engine efficiency. It has been determined, through use of the performance model, that if droplet sizes can be reduced to one micron, the goal can be realized. The performance of a full admission turbine with 1 micron droplets is indicated by the upper curve of Figure 33. Again, the rotor and blade geometry is identical to that tested. The lower bound of the range was calculated using the measured cascade vapor velocity coefficient, while the upper bound was calculated on the basis of pure vapor turbine velocity coefficient [36]. The uncertainty associated with liquid pumping losses applies to this case, just as it did to the full admission calculation presented previously. The performance increase afforded by neglecting this loss is not so significant in the present case (dashed lines), as in the previous case where all the droplets were assumed to impact the blades.

In this case, the fraction of droplets impacting the blades is much smaller, reducing the mass flow rate of liquid that must be removed from the blades.

Though a nozzle coefficient of 0.94 was assumed for this case, it is very probable that successful droplet size reduction will also significantly increase the thrust coefficient providing the possibility of further gains in efficiency. It should also be noted that as droplet size is reduced, the fluid mixture will begin to behave as a dense gas. Shock phenomena will become more severe compared to the case with larger droplets.

The advanced design operates at a higher blade speed ratio than the less efficient designs. This reduces the relative velocity of the fluid mixture in the blade channel. It is calculated that the velocities entering the blade passage will be slightly subsonic relative to the vapor. The mixture relative velocity will be considerably above the two-phase mixture choking velocity. As the droplet diameter is reduced to 1 micron severe shock phenomena can occur under these conditions. Hence, reducing the droplet diameter to 1 micron allows more droplets to pass through the blade passages without impact so that the 70 percent research goal can be reached, but at the same time the probability of occurrence of shock phenomena is also increased. It is thus essential to have a two-dimensional, two-phase flow calculational capability which can include shock wave phenomena in order to assure realization of the high efficiency designs sought. Another analytical tool in the area of blade design is also

required. Presently, a two-phase boundary layer model does not exist. This too will be essential to high performance blade design.

Figure 34 shows the dramatic change in engine efficiency with droplet diameter. The portion of liquid captured is also shown as a function of droplet diameter. The portion of liquid captured does not equal 100 percent, even for large droplets, because some evaporation is allowed to occur upon droplet impact in the performance model.

Another possible approach to improving performance would seem to be the increase of blade width, and hence blade channel radius of curvature and the blade spacing. In order to investigate this, a special version of the performance model was set up to examine what gains might be made in this manner. As with the previous model, blades were assumed to be defined by circular arcs and straight lines. Only pure impulse blading was considered. Blade spacing was varied as a function of blade width according to an optimum blade spacing criteria [22] used for vapor turbines, see Appendix 5, (5.1). It is not suggested that these conditions will represent optimum spacing for the two-phase mixture, but the approach does lead to some interesting results which assist in understanding the bounds of the problem. A vapor velocity loss coefficient, see Appendix 5, (5.38), was used which was developed from vapor turbine correlations [22] and is slightly more optimistic than the one used for the upper bound of the 1 micron results for Figure 33. This particular loss coefficient formulation was used here because it included blade geometry variation. The nozzle performance for the calculations was assumed to be ideal, with no slip and no losses. The

Fig. 34. Engine efficiency and portion of liquid captured as a function of droplet diameter for full admission.

blades were assumed to be shrouded with no lap. The performance for a given droplet size and blade width was optimized by adjusting the blade turning angle. These results should not be used to estimate real machine performance, since they are somewhat idealized; rather, they should be thought of in relative terms for the indication of trends.

Idealized rotor efficiency is shown as a function of droplet size for three different blade widths in Figure 35. The lowest case (dashed line) indicates the performance for a fixed turning angle (for reference), using the same blade turning angle and blade width as used for the dynamic blade test, but with blade spacing as prescribed by the model. The lower solid curve is for the same width blade, but the turning angle is adjusted for optimum performance. The two other curves are for blade widths of two times and three times the blade widths used in the dynamic blade test, as indicated. Even with blade widths as large as 30 cm, performance is not improved if the droplets are not smaller than 6.5 microns. For 3 to 4 micron droplets, the larger blade widths do improve performance. Thus, widening the blades and increasing the blade spacing may allow the achievement of the 70 percent engine efficiency research goal with droplets larger than 1 micron.

It should be emphasized that the present performance model is not fully coupled. Hence, the results presented in Figure 35 involve some uncertainty. A fully coupled two-dimensional blade flow calculation technique is required to properly calculate the dependence of

Fig. 35. Idealized rotor performance as a function of droplet-diameter for various blade widths.

performance upon blade spacing, curvature, turning angle, and droplet size.

For non-integral blade designs, the wide blades will require increased working stresses in the blade attachment region. This further complicates the design and may, aside from fluid dynamic considerations, place constraints on the maximum blade width.

In summary, extrapolations to full admission which assumed that the measured performance of the nozzle alone can be achieved with proper spacing of the nozzle and rotor and removal of the shroud, predicted a range of performances between 38 and 48 percent. Though uncertainty is involved in estimating the contribution of liquid pumping loss to the performance in the full admission case, a two-dimensional liquid layer approximation indicated that for large droplets, the expected performance will be 38 to 41 percent. The 70 percent engine efficiency research goal appears to be credible. Its achievement, with the rotor and blade geometry tested, will require droplet size reduction to one micron or less. Increases of blade width and blade spacing improve performance and increase the size of the droplet which will pass through the blade channel without impact. Finally, a two-dimensional two-phase flow calculational model will be required for the design of high performance blades and nozzles in order to achieve the research goal.

Recommendations For Future Effort

It is imperative that droplet size and size spectra be measured experimentally [40]. Such measurements can form the basis for more accurate analytical predictions and further understanding of the boundaries of the problem. This information is vital for the design of advanced two-phase expanders in the effort to achieve the research goal. A two-dimensional two-phase calculational technique is presently being developed so that more efficient blades and nozzles can be designed. It is very important that this work be completed to assure achievement of the research goal since two-phase shock behavior may become considerably more important to performance as droplet diameter is reduced to 1 micron. A two-phase boundary layer code to include the effects of droplets should be developed to enhance the two-dimensional calculational model. In addition to these two basic areas, nonequilibrium effects should be included in 1D2FAZ to allow studies of droplet break-up in single component flow due to boiling phenomena [41]. This should provide a more reliable criteria than that of the Weber number for single-component two-phase flows. Nonequilibrium effects should be included in the two-dimensional code if it appears necessary and feasible based upon the one-dimensional nonequilibrium results. Additional two-phase flow measurement techniques must be developed for determination of local droplet and vapor velocities. The stabilities of the liquid layer, including surface tension effects on the blade surface, should be investigated to determine its behavior so that the liquid pumping losses can be better defined for full admission when droplets are large enough to impact the blades.

Surface tension control should be investigated, since it is an important parameter to both the Weber number and the boiling droplet break-up criteria. The distribution of two-phase fluid mixtures to many nozzles, as will be required in a full admission design, should be studied experimentally to ascertain that there are no surprises. Material studies presently being carried on should be continued in order to decide upon a suitable material for the chemical environment as well as one that will withstand the stress and erosion conditions of the rotor and blades. It appears that brine scaling can be controlled [19], but continued effort should be concentrated on the specific application of scale control to the Total Flow turbine.

Conclusion

A prototype two-phase impulse expander has been selected, designed and built for testing in the laboratory. The expander has been successfully tested in the laboratory. A performance model, together with nozzle characterizations based upon numerical and experimental techniques, and static cascade results have been combined to produce performance predictions which agree well with measured performance data. The performance model was used to extrapolate to full admission performance based upon previously measured nozzle performance. Droplet size reduction has been identified as being crucial to improved performance. The development and implementation of a fully coupled two-dimensional two-phase design calculational technique capable of predicting shock behavior is also necessary to maximize the performance

of an advanced design. It appears probable that improved performance will result from designs using wider blades and blade spacing, even for droplets larger than one micron. This conclusion requires verification by the coupled two-dimensional two-phase flow calculations. Finally, it has been determined that if droplet size is reduced to one micron or less and the two-phase calculational techniques are developed and employed for two-dimensional design of blades and nozzles, the research goal of 70 percent engine efficiency can be achieved with a blade width and turning angle equal to that of the blade profile tested.

Acknowledgements

I take this opportunity to thank many of the people who have helped to make this effort possible. First, I thank my wife, Mary, who perservered throughout the study leading to this dissertation. As if that wasn't enough, she then had the fortitude to edit most of what has been written herein. Special thanks to my advisors: Professors C. W. Beadle, C. T. Crowe, and W. H. Giedt. These gentlemen have been real gems—providing me with encouragement, assistance, and a wonderful breadth of insight. C. C. Gardiner designed much of the hardware associated with the expander and dynamometer. His experience, attention to detail, and hard work were a major contribution throughout the conceptual design, fabrication, and debugging of the equipment. T. W. Alger provided a great deal of nozzle design, analysis, and testing experience plus many very fruitful technical discussions. P. A. House contributed both creativity and effort, particularly during debugging and testing. G. W. Wright designed and operated the hydraulic dynamometer/drive and the control system for the expander testing. H. Weiss provided insight through his materials experience by assisting in the specification of material processing and in property determination. H. J. Wiesner, W. P. Frey, and W. Kelley worked wonders in applying the electroless nickel plating of the rotor. L. F. Meisner, R. T. Hasbrouck, and N. L. Rector gave of their expertise in designing and understanding the recording equipment and instrumentation, in calibrating and operating said equipment, and in helping to solve mysteries associated with the same. Consultants: Dr. O. E. Balje and Professor R. F. Steidel were helpful in providing

suggestions concerning turbomachine design and vibrations. G. H. Lathrop, L. L. Jackson, C. W. Hall, and D. L. Podesta disassembled, improved, and reassembled the expander and dynamometer more times than it is appropriate to remember. A. A. Anderson very competently smoothed out many of the bumps and significantly reduced the wear and tear on all of us. C. M. Gordon provided many helpful discussions during the conceptual stages of this effort, as well as some very clever demonstration models. Thanks to G. C. Dennis for her outstanding handling of the Mechanical Engineering Education office. G. L. Dittman helped me many times as the Total Flow Turbine group leader through his guidance, encouragement, and ability to separate the wheat from the chaff. Finally, I thank A. W. Lundberg and A. L. Austin for the management and leadership exhibited in their direction of the geothermal conversion effort, and for their assistance to me.

References

1. Austin, A. L., Higgins, G. H., Howard, J. H., "The Total Flow Concept For Recovery of Energy from Geothermal Hot Brine Deposits", UCRL-51366, April 1973, Lawrence Livermore Laboratory, Livermore, California.
2. Austin, A. L., "Prospects for Advances in Energy Conversion Technology for Geothermal Energy Production", Proceedings of the Second United Nations Symposium on the Development and Use of Geothermal Resources, San Francisco, U. S. A., May 20-29, 1975, Vol. 3, pp. 1925-1937.
3. Stodola, A., Lowenstein, L. C., Steam and Gas Turbines, McGraw-Hill Book Co. Inc., New York, 1927.
4. Varljen, T. C., "The transport of Atomized Drops in Wet Vapor Turbines", WANL-TME-1836, August 5, Westinghouse Electric Corporation.
5. Kirillov, I. I., Yablonik, R. M., Fundamentals of the Theory of Turbines Operating on Wet Steam, 1968, Translation of Osnovy teorii vlazhnoparovykh turbin, Mashinostroyeniye Press, Leningrad.
6. Elliott, D. G., Hays, L. G., "Two-Phase Turbine Engines", September 1976, Eleventh Intersociety Energy Conversion Engineering Conference.
7. Comfort, W. J., Alger, T. W., Giedt, W. H., Crowe, C. T., "Calculation of Two-Phase Dispersed Droplet-In-Vapor Flows Including Normal Shock Waves", A. S. M. E. Paper No. 76-WA/FE-31, Presented at the A. S. M. E. Winter Annual Meeting, December 5, 1976.
8. Weiss, H., R. F. Steidel, A. W. Lundberg, "Performance Test of a Lysholm Engine", UCRL-51861, July 1975, Lawrence Livermore Laboratory, Livermore, California.
9. House, P. A., "Helical-Rotor Expander Applications for Geothermal Energy Conversion, UCRL-52043, April 1976, Lawrence Livermore Laboratory, Livermore, California.
10. Elliott, D. G., "Comparison of Brine Production Methods and Conversion Processes for Geothermal Electric Power Production" EQL Report No. 10, July 1975, Environmental Quality Laboratory, California Institute of Technology, Pasadena, California.
11. Alger, T. W., "The Performance of Two-Phase Nozzles for Total Flow Geothermal Impulse Turbines", Proceedings of the Second United Nations Symposium on the Development and Use of Geothermal Resources, San Francisco, U. S. A., May 20-29, 1975, Vol. 3, pp. 1889-1895.
12. Crowe, C. T., Sharama, M. P., Stock, D. E., "The Particle-Source-in-Cell (PSI-Cell) Model For Gas-Droplet Flows", A. S. M. E. Paper No. 75-WA/HT-25.

13. Forsnes, V. G., Ulrich, R. D., "A Literature Review and Discussion of Liquid Particle Breakup in Gas Streams, NWC TP4589, July 1968, Naval Weapons Center, China Lake, California.
14. Crowe, C. T. is developing a boiling breakup criteria for single component flows, private communication.
15. Alger, T. W., private communication.
16. Balje, O. E., Binsley, R. L., "Final Report, Performance Prediction: Optimization Using Flyid Dynamic Criteria", R-6805, December 1966, Rocketdyne.
17. Csanady, G. T., Theory of Turbomachines, McGraw Hill Book Co., 1964.
18. Weiss, H., "Geothermal Two-Phase Flow Test Facility", Proceedings of the Second United Nations Symposium on the Development and Use of Geothermal Resources, San Francisco, May 20-29, 1975, Vol. 3, pp. 2065-2068.
19. Austin, A. L., etal. Lawrence Livermore Laboratory Geothermal Program Status Report (to be published in April 1977)
20. Alger, T. W. designed and tested the nozzle for the dynamic blade test, private communication.
21. Goldberg A., "Quarterly Report, July-September 1976, Geothermal Materials Studies", UCID-17261-76-3, September 1976, Lawrence Livermore Laboratory, Livermore, California.
22. Horlock, J. H., Axial Flow Turbines, Butterworth and Company Ltd., 1966.
23. Elliott, D. G., Weinberg, E., "Acceleration of Liquids in Two-Phase Nozzles", NASA Technical Report 32-987, July 1968, Jet Propulsion Laboratory, California Institute of Technology, Pasadena California.
24. Weiss, H., private communication.
25. Weisner, J. W., Comfort, W. J. III, Frey, W. P., Gardiner, C. C. Jr., Kelley, W. K., "Electroless-Nickel Plating of a Large Turbine Rotor for the Geothermal Program", UCID-17253, September 1976, Lawrence Livermore Laboratory, Livermore, California.
26. Goudreau, G. L., DTVIS2/POST2, Users Manual, Lawrence Livermore Laboratory Internal Memorandum, June 19, 1972.
27. Kroon, R. P., "Turbine-Blade Vibration Due to Partial Admission", Journal of Applied Mechanics, December 1940.
28. Weiss, H., private communication.

29. Alger, T. W. designed the nozzle for the static cascade test, private communication.
30. Crowe, C. T., "Conservation Equations for Vapor-Droplet Flows Including Boundary-Droplet Effects", UCRL 521821, December 3, 1976, Lawrence Livermore Laboratory, Livermore, California.
31. Crowe, C. T., private communication.
32. Crowe, C. T., private communication.
33. Crowe, C. T., private communication.
34. Radcliffe, C. W., Comfort, W. J., "Constrained Minimization of a Function of Many Variables, UCRL-51517, January 24, 1974, Lawrence Livermore Laboratory, Livermore, California.
35. Crowe, C. T., Interpretation of Static Pressure Distribution on Static Blade Tests, Lawrence Livermore Laboratory Internal Memorandum, November 22, 1976.
36. Vavra, M. H., Aero-Thermodynamics and Flow in Turbomachines, Robert E. Krieger Publishing Co., Huntington New York, 1974.
37. Warda, H. A., Mobbs, F. R., Cole, B. N., "Gas Solids Flow in Supersonic Nozzles", A. S. M. E. Paper no. 75-WA/HT-35.
38. Wright, G. W., Operational Instructions and Procedures for Turbine Hydraulic Drive/Brake System, ENR-77-5, March 15, 1977, Lawrence Livermore Laboratory, Livermore, California.
39. Alger, T. W., private communication.
40. Alger, T. W. is presently developing experimental methods for droplet velocity and size spectrum measurements, private communication.
41. Crowe, C. T. is modifying 1D2FAZ to study temperature nonequilibrium phenomena leading to droplet boiling in single component flows, private communication.
42. House, P. A., private communication.
43. House, P. A., "Analysis of the Radial Outflow Reaction Turbine and Application in Geothermal Advanced Flashed Steam Conversion Systems", ENE 76-27, October 26, 1976, Lawrence Livermore Laboratory, Livermore, California.
44. Powell, M. J. D., "A Method for Minimizing a Sum of Squares of a Nonlinear Function Without Calculating Derivatives", Computation J., Vol. 7, pp. 303-307, 1965.
45. Hindmarsh, A. C., "GEAR Ordinary Differential Equation System Solver", UCID-30001 (Rev. 3), December 1974, Lawrence Livermore Laboratory, Livermore, California.

46. Leonard, L., "ZEROIN--A Subroutine to Calculate One Real Zero of a Real Continuous Function", UCID-30092, April 1974, Lawrence Livermore Laboratory, Livermore, California.

47. Mann, R. W., Marston, C. H. "Friction Drag on Bladed Disks in Housings as a Function of Reynolds Number, Axial and Radial Clearance, and Blade Aspect Ratio and Solidity", Journal of Basic Engineering, December 1961.

48. Roelke, R. J., "Miscellaneous Losses" from the publication: Turbine Design and Application, Vol. II, NASA SP-290, 1973.

APPENDIX 1

Nomenclature

- A = area normal to the flow direction
 B = blade chord width
 b_w = blade spacing
 C = experimentally determined normalization factor for disk friction and windage
 c = choking velocity, specific heat if subscripted
 C_D = drag coefficient
 C_f = skin friction coefficient
 C_M = moment coefficient
 C_T = thrust coefficient
 D = droplet diameter
 E = modulus of elasticity
 F = force
 G = disk friction constant
 H = blade height
 h = specific enthalpy
 K = proportionality constant
 k = thermal conductivity
 K' = proportionality constant
 KE = kinetic energy per unit mass
 M = moment
 $\dot{m}$ = mass flow rate
 N = number of droplets per unit volume

n = number of cycles

N = Nusselt number

P = power

p = pressure

R = vapor mass flow rate ratio

r = radius

R = Reynolds number

S = slip ratio (droplet velocity/vapor velocity), mass flow source if subscripted

s = source of mass flow per unit surface area

SF = safety factor

T = temperature

U = blade velocity

u = radial displacement

V = velocity

W = work

w = work per unit mass

$\dot{W}$ = work per unit time (power)

We = Weber number

X = vapor mass fraction

x = coordinate

y = coordinate, or displacement (amplitude) if subscripted

z = coordinate

α = coefficient of thermal expansion or enthalpy correction

β = blade angle (with plane of rotor, Figure 5.1)

γ = incidence angle of droplet, or radius ratio (liquid nozzle to vapor nozzle)

- δ = logarithmic decrement or local liquid boundary layer thickness or flow deflection angle (oblique shock)
 ϵ = void fraction
 ζ = disk friction and windage exponent
 η = efficiency
 θ = angle of admission
 θ = oblique shock wave angle or nozzle angle or coordinate direction, or as in Figure 5.1
 λ = relative velocity coefficient for blades (exit/entrance)
 μ = viscosity
 ξ = energy loss coefficient
 ρ = density
 σ = stress (normal)
 τ = stress (shear), time constant if unsubscripted
 ϕ = blade turning angle (Figure 5.1)
 ψ = angle between blade chord direction and radius vector (Figure 5.3)
 Ω = blade speed ratio (blade speed/isentropic nozzle velocity)
 ω = angular velocity

Subscripts

- a = actual (measured)
 al = alternating
 B = refers to conditions at blade surface
 C = Coriolis
 cn = captured on nozzle
 D = disk friction and windage
 e = exit without impact

en = endurance

f = radius of the blade corresponding to the outermost portion of the nozzle

h = evaporation resulting from frictional vapor enthalpy increase

I = initial

i = inside or root

IM = impact

is = isentropic

j = index

KE = kinetic energy

L = liquid

m = mean

mx = maximum

o = outside or reference

P = pumping, or pressure (referring to pressure surface of blade)

pa = partial admission

R = relative

r = radial

s = sticking (or adhering), or suction (referring to suction surface of blade)

st = step (square pulse) amplitude

V = vapor

y = yield

1 = entering side of control volume, except as in Figure 3.6

2 = exiting side of control volume, except as in Figure 3.6

APPENDIX 2

The Hero or Radial Outflow Turbine

The most often referred to "savior" for two-phase geothermal resource conversion machines is the Hero turbine. It is also a machine which is frequently analyzed incorrectly. Before proceeding further, the performance characteristics for a Hero turbine will be derived to provide insight into the suitability of such a machine for two-phase flow.

A single phase flow analysis is performed for simplicity. Two-phase considerations can be discussed with reference to the single phase analysis. The nozzles will be assumed to be very short to simplify the geometry, Figure 2.1, so that the effects of forces on the fluid within the nozzle may be neglected.

In order to establish a reference velocity, the machine is first imagined to be blocked such that it cannot rotate. The velocity of the fluid exiting the nozzle under these circumstances is defined as V_{is} , and its magnitude results from ideal conversion of the potential energy in the fluid to kinetic energy. For later reference, V_{is} for the particular resource considered (Table 1) is approximately 727 m/s.

The force tending to rotate the nozzle at velocity U is the product of the mass flow rate of the fluid and the absolute velocity as shown in

Fig. 2.1. Idealized hero turbine.

(2.1). The exit velocity of the fluid relative to the nozzle, V_e , is as yet undefined except when the nozzle is at rest and is defined as V_{is} . The ideal power development for such a turbine is the product of the velocity of the nozzle and the nozzle thrust.

$$F = \dot{m}(V_e - U) \quad (2.1)$$

$$P = \dot{m}U(V_e - U) \quad (2.2)$$

Consider the fluid as it passes from the axis of the machine to the rotating nozzle. Work is done on the fluid by the rotor in two ways.

1) The fluid is accelerated from its small entrance velocity to the nozzle velocity U . The work required to accomplish this is the change in kinetic energy:

$$w_{KE} = U^2/2 \quad (2.3)$$

2) The fluid is also compressed as it moves radially outward. The potential energy, or pump work, is derived below, referring to Figure 2.2.

$$(p + dp - p)A = r\omega^2 dm$$

$$dp = \rho r \omega^2 dr$$

$$dp/\rho = r\omega^2 dr \quad (2.4)$$

Fig. 2.2. Radial equilibrium for fluid element.

the relationship in (2.4) applies to both compressible and incompressible isentropic compression. Thus, the pump work for either compressible or incompressible flow can be idealized as shown in (2.5).

$$w_p = U^2/2 \quad (2.5)$$

The force on the fluid, exerted by the rotor, which produces the kinetic energy and compression is the Coriolis force. The work done by the rotor on the fluid through the Coriolis force equals the sum of the change in kinetic energy and pump work as shown below, referring to Figure 2.3.

$$\begin{aligned} dF &= 2\omega r dm \\ dF &= 2\omega m dr \\ dM &= r dF \\ d\dot{W}_C &= \omega dM \\ d\dot{W}_C &= 2\omega^2 m r dr \\ \dot{W}_C &= \dot{m} U^2 \\ w_C &= U^2 \end{aligned} \quad (2.6)$$

Fig. 2.3. Tangential equilibrium for fluid element.

In the relative frame (traveling with the nozzle), only the pump work adds to the energy available to accelerate the fluid through the nozzle. The kinetic energy change is required to cause the fluid to appear in the nozzle at the nozzle velocity. Assuming an isentropic nozzle expansion, it is now possible to determine the fluid velocity exiting the nozzle (in the relative frame) since its value must be the sum of the potential energies associated with the static fluid and with the compression due to rotation. The inclusion of the pump work as part of the potential energy available to accelerate the fluid across the nozzle is what is often neglected in erroneous derivations of the performance characteristics of the Hero turbine.

$$V_e^2 = U^2 + V_{is}^2 \quad (2.7)$$

Substituting the value for V_e (2.7) into (2.2) establishes the ideal power output in terms of blade speed and the static nozzle exit velocity V_{is} .

$$P = \dot{m}U(\sqrt{V_{is}^2 + U^2} - U) \quad (2.8)$$

Equation (2.8) may be rewritten in terms of efficiency η , blade speed ratio β , and the nozzle thrust coefficient C_T as follows.

Fig. 2.4. Single phase ideal performance for a hero turbine for various thrust coefficients.

$$\eta = 2P/\dot{m}V_{is}^2 \quad 0 < \eta < 1$$

$$\Omega = U/V_{is} \quad 0 < \Omega < \infty$$

$$C_T = V_a/V_e \quad 0 < C_T < 1$$

$$\eta = 2\Omega(C_T \sqrt{1 + \Omega^2} - \Omega) \quad (2.9)$$

The resultant expression is shown in (2.9) and is plotted in Figure 2.4 for various nozzle thrust coefficients. In an ideal case, ($C_T = 1$) one can observe that the efficiency approaches 100 percent at infinite blade speed. This characteristic of the pure reaction machine leads to impractical stress requirements. Even at lower thrust coefficients, the peak performance points occur at blade speeds which are exceedingly high for the particular resource considered here. For reference, the best two-phase nozzle thrust coefficient measured to date is approximately 0.94.

With this background information, the two-phase applications may now be discussed. The operating state points (Table 1, text) are assumed. Two possible methods for handling the fluid exist: 1) pass the two-phase mixture through a single passage and nozzle, and 2) separate the mixture into liquid and vapor before entering the machine and then provide separate passages and separate nozzles for each.

The ideal thermodynamic processes for the two-phase mixture will now be

presented in terms of the temperature entropy diagram for the two fluid handling schemes described (Figure 2.5). In the first case, the mixture is compressed from initial state a to state b by centripital acceleration and then expanded in the nozzle from states b to c. In the second case, an ideal separation of liquid and vapor occurs from initial state a to states g and d on the saturated liquid and saturated vapor lines respectively. The fraction of the total mass flow rate associated with the vapor is R and with the liquid is $(1-R)$. The separated fluids are then compressed from states g to h into the compressed liquid region and d to e into the superheated vapor region by centripetal acceleration. As the liquid is expanded through its nozzle (h to i) it flashes to a two-phase mixture at g. As the vapor is expanded through its nozzle (e to f) it begins to condense to a two-phase mixture at d.

When one examines the first case in more detail, one must first consider what determines the state at b. If the two-phase mixture behaves as a continuum, meaning that the droplets are very small (much less than 1 micron), then the pressure at b is proportional to the mixture density. If this was true, the machine would perform in the same manner as for a single phase fluid. Small droplets would not be expected since no large negative pressure gradients occur at the entrance to the rotor or in the radial passages. Because the droplets are relatively large, they are removed from the mixture by Coriolis acceleration and form a liquid rivulet on the wall of the passage. The pressure is thus proportional to the continuum or saturated vapor phase density (assuming thermal equilibrium between phases). It can be

Fig. 2.5. Ideal process descriptions in terms of the temperature entropy diagram for water.

argued that if the liquid was inviscid, it could gain kinetic energy by being free to accelerate radially outward, in which case the kinetic energy gain would ideally be equal to the pump work as shown in the following derivation:

$$\begin{aligned}
 \ddot{r} - \omega^2 r &= 0 \\
 \ddot{r} &= \dot{r} \frac{d\dot{r}}{dr} \\
 \frac{d(r^2)}{dr} &= 2\omega^2 r \\
 \dot{r}^2 &= U^2 \\
 \Delta KE &= U^2/2
 \end{aligned}
 \tag{2.10}$$

Assuming no losses in the nozzle, this could result in the ideal performance curve for $C = 1$ in Figure 2.4. Because of the relatively high viscosity of water and the high void fraction flow (indicating that liquid rivulets will be very thin) considered, it seems highly improbable that the kinetic energy gain for the liquid through radial acceleration will be significant. The possible gain in kinetic energy of the liquid through radial acceleration is thus neglected. The enthalpy change between states a and b can be calculated numerically by integrating the pressure gradient along the rotating column using saturated equation of state relations. The continuum phase density is assumed to be that of the saturated vapor. The results (for this specific case) are shown (Figure 2.6) as a ratio, α , of the enthalpy derived from assuming the vapor is the continuum phase to that determined from assuming that the mixture is the continuum phase as a function of the blade speed ratio.

Fig. 2.6. Enthalpy correction as a function of blade speed ratio.

Fig. 2.7. Two-phase mixture hero turbine.

Ideally, assuming that the mixture can be accelerated isentropically, ignoring slip and friction effects, a new exit velocity for the two-phase mixture can be defined and substituted into (2.8).

$$\eta = 2\Omega(C_T \sqrt{1 + \alpha\Omega^2} - \Omega) \quad (2.11)$$

Here, V_{is} is defined by the enthalpy associated with the process a to c. The results for various thrust coefficients are presented in Figure 2.7. Rather low performance results, even when an optimistic thrust coefficient is applied. The optimum blade speed is approaching the limit considered to be acceptable from strength considerations.

Returning now to the second case, where the liquid and vapor are treated separately, all of the compression work can be attained for each phase. New ideal nozzle velocities, V_{Lis} and V_{Vis} can be defined, based upon the isentropic enthalpy differences between states g and i and states d and f. The mass averaged isentropic velocity may be defined as:

$$V_{is}^2 = V_{Lis}^2 (1 - R) + V_{Vis}^2 R \quad (2.12)$$

and must be equal to the isentropic velocity defined by the process a to c. The expression for the power output of the machine may now be written:

$$P = \dot{m}U[R(\sqrt{V_{Vis}^2 + U^2} - U) + (1 - R)(\sqrt{V_{Lis}^2 + U^2} - U)] \quad (2.13)$$

Introducing thrust coefficients (Cr_V for vapor and Cr_L for liquid) and expressing the results as an efficiency:

$$\eta = 2U/V_{is}^2 [R(Cr_V \sqrt{V_{vis}^2 + U^2} - U) + (1 - R)(Cr_L \sqrt{V_{lis}^2 + U^2} - U)] \quad (2.14)$$

Efficiency may be plotted as a function of blade speed, or as has been done, as a function of the blade speed ratio. Using a very optimistic thrust coefficient for the liquid fraction of 0.8 and 0.94 for the vapor fraction, the results are not promising (Figure 2.8), though the blade speed is structurally possible.

If one assumes that the liquid nozzle is a different radius from the axis of rotation than the vapor nozzle, as suggested by House [42], (2.14) can be modified as shown in (2.15).

$$\eta = 2U/V_{is}^2 [R(Cr_V \sqrt{V_{vis}^2 + U^2} - U) + \gamma(1 - R)(Cr_L \sqrt{V_{lis}^2 + \gamma^2 U^2} - \gamma U)] \quad (2.15)$$

The optimum value of γ , the ratio of the nozzle rotation radii, is dependent upon the assumed thrust coefficients for a given resource and must be determined. The expression for η (2.15) was maximized numerically with respect to the two variables U and γ , using POON [34]. The results for the previously introduced values of the thrust coefficients are shown in Figure 2.9. Modest gains have been made in performance, compared to the case for $\gamma = 1$ (Figure 2.8), but at the

Fig. 2.8. Separated liquid and vapor two-phase turbine-nozzles at same distance from the axis of rotation.

Fig. 2.9. Separated liquid and vapor two-phase turbine nozzles at optimum distances from the axis of rotation.

cost of blade speeds which are structurally impossible for this resource.

Discussion

The sensitivity of the performance to the thrust coefficient(s), and the high blade speeds required for optimum performance for the resource considered here, were the reasons originally used to eliminate this type of machine from the list of candidate expanders. This conclusion can be drawn reasonably well from the single phase performance but the additional details have been included here for two-phase expansion.

It should be emphasized that the performances indicated are definitely upper bounds. For standard converging-diverging two-phase nozzles, the thrust coefficients used have been optimistic. Slip between phases as well as rotational body force effects on nozzle performance have been neglected. Even if one were to design for a given rotor speed by contouring the nozzle and duct(s) leading to the nozzle(s), the performance would not exceed the cases presented. Additional losses associated with disk friction and liquid clinging to the rotor can be quite significant. Actual rotor geometry, which would introduce additional loss and complexity have also been ignored by assuming very short nozzles.

The one variation not addressed, which should be mentioned briefly in light of renewed interest in this machine for lower enthalpy resources,

concerns the second case where the liquid and vapor were separated prior to entering the expander. House [43] has proposed that the steam fraction be used in a conventional manner while a radial outflow turbine might be used to expand the saturated liquid. Here, blade speeds can be structurally acceptable. A special two-step nozzle has been designed to increase nozzle performance for compressed liquid entrance conditions and to reduce the possibility of liquid interaction with the nozzle wall.

APPENDIX 3

Structural Considerations

Finite Element Modeling of the Rotor

Finite element calculations were made to establish the adequacy of the design of the aluminum rotor. The two-dimensional analysis was chosen as adequate, rather than a three-dimensional approach. Had an attachable blade design been used, the three dimensional analysis would have been more important.

Figure 3.1 shows the aluminum disk and steel hub assembly as well as the elements into which they were divided for calculational purposes. Application of the two-dimensional approach to the axisymmetric rotor assembly made it necessary to model the presence of the blades in the third dimension at the outer boundary (blade/disk interface) of the disk. The blades, shown in Figure 3.2, may be thought of as constraining the outer surface of the disk to remain parallel to the axis or cylindrical. The cantilevered rim below the blades would deflect more at the outer edge than at the plane of symmetry if a uniform axial pressure distribution was applied at the blade/disk interface to account for the inertial loading represented by the blades. The average density of the blade and blade passage combined may be approximated to have an axial distribution as shown in Figure 3.3. The average pressure distribution representing the blades would have this

Fig. 3.1. Elements used for calculation.

Fig. 3.2. View of blades/rotor.

Fig. 3.3. Idealized effective density distribution for blade loading characterization.

same shape. By applying this pressure distribution to the disk/blade interface, it is found that the interface deflects uniformly. Thus the boundary condition is modeled, insofar as the change in section below the blades is concerned. Stresses at the disk/blade interface are not correctly handled in this fashion and were treated separately with appropriate stress concentration factors.

The interference fit of the rotor and hub was studied carefully. In order to move the bearings as close to the plane of symmetry of the rotor as possible, the somewhat complex steel hub geometry required the use of the finite element analysis for this region. The axial interface (parallel to the symmetry plane) between the aluminum rotor and steel hub was nodally connected in the region where the bolts were inserted. Below this region, the hub and disk were not connected and some separation occurs (Figure 3.4). As may be observed, some axial compression occurs in the bolted region because of the assumption of no sliding at the (shrink fit) cylindrical interface between the hub and rotor. An indication of the tendency of the steel and aluminum interface to separate was observed near the lower radial corner of the bolted region as indicated by the tensile stress (Figure 3.4). Increased radial interference, compared to the initial static, room temperature assembly condition, results during operation from both the difference in thermal expansion of the steel hub and aluminum rotor and from the tendency of axial contraction and radial growth from rotation. It was important to see that stresses in either the hub or the rotor were not excessive under all possible operating conditions.

Fig. 3.4. Axial stress in the hub region (displacements multiplied by 50)

Fig. 3.5a. Maximum principal stresses in the aluminum rotor (displacements multiplied by 50)

Fig. 3.5b. Maximum principal stresses in the hub region (displacements multiplied by 50)

The poor heat conduction properties of stainless steel lead to an additional concern that a transient thermal conditon might exist during warm-up of the machine such that a more severe stress than the final running stresses might result. The total surface of the rotor and hub as seen in Figure 3.1 was held at constant temperature to simulate the presence of steam during warm-up, and the transient coupled heat conduction/thermal stress problem was calculated using DIVIS2. This showed that the maximum stresses still occured at the final steady-state condition.

Contour plots of the maximum principal stresses are shown for the aluminum rotor in Figure 3.5a and for the hub region for both the steel and the aluminum in Figure 3.5b. More detailed analysis was not undertaken since maximum stresses throughout the assembly were low compared to the yield strengths.

Blade Vibration

A pessimistic analysis of the effects of partial admission on blade vibration was performed by Kroon [27]. The analysis assumes that the passing nozzle provides a square pulse forcing function on the blades which always occurs in integral half periods of the blade natural frequency. A square pulse is certainly conservative but is assumed for simplicity. In addition, the pulses are always assumed to be timed so that maximum amplification occurs. Physically, this leads to an improbable situation though one must agree it could occur. Kroon also

assumed that the material damping coefficients represented the structural damping. This appears, from measurements made in support of this effort, to be conservative.

It was found from damping measurements, originally on blade sections machined for experience prior to machining the rotor and later on the rotor, that the damping could be idealized as having three decrements. The measurements and the decrements used will be presented later in this Appendix.

Kroon included cases for large arcs of partial admission. In the single nozzle considered here only 12 degrees of admission occurs. This small arc permits one to neglect the damping during the square pulse application. Otherwise, using the same general theory that Kroon developed, the amplification factor is derived as follows (referring to Figure 3.6).

$$y_{mx} = y_d + y_{st}$$

$$y_d = y_I + y_{st}$$

$$y_{mx} = y_I + 2y_{st}$$

$$y_I = y_{mx} e^{-[n_1(\delta_1 - \delta_2) + (n_2(\delta_2 - \delta_3) + n_3\delta_3)]} \quad (n > n_2)$$

$$\frac{y_{mx}}{y_{st}} = \frac{2}{1 - e^{-\sum n_j(\delta_j - \delta_{j+1})}} \quad (j \leq 2) \quad (3.1)$$

Fig. 3.6. Amplification factor derivation relationships

A simple linear relationship is assumed for the steady state force on the blade versus blade speed ratio (which is true in an ideal impulse turbine and is conservative).

$$F_{st} = K(1 - \Omega) \quad (3.2)$$

Nominal bending stresses near the blade root were estimated using beam theory, and modified with appropriate stress concentration factors for the design. As an additional check, strain gages were installed in critical areas and static loading was applied to corroborate the calculated stresses. The maximum steady state bending stress occurring at the blade root was then modified by the amplification factor (3.1) over the operating speed range of the turbine (3.2).

$$\sigma_{mx} = K'(1 - \Omega)(y_{mx}/y_{st}) \quad (3.3)$$

The results of this calculation are presented in Figure 3.7. The amplification factor derivation assumed only the worst case of multiple one-half period pulses. Thus, the results shown in Figure 3.7 represent an upper bound and are only approximately correct at 4,800 rpm and 2,800 rpm where even half-period excitation is approximated. Even at these points, the probability of exact reinforcement each revolution is very small.

Basic endurance strength data for 7075 T-6 aluminum was modified for surface, size, reliability, and temperature. This reduced the

Fig. 3.7. Alternating and mean (centripetal) stress for critical blade attachment region as a function of rotor speed.

Fig. 3.8. Factor of safety for critical blade attachment region as a function of rotor speed based.

endurance strength for the design by a factor of 1.73 to 71.7 MPa. By applying the Soderberg law, one can determine the factor of safety available in the design by comparing the calculated alternating and mean stresses with the design endurance strength. The Soderberg law, including the safety factor, may be written:

$$SF = \frac{\sigma_{en}}{\sigma_{al} + \sigma_m \sigma_{en} / \sigma_y} \quad (3.4)$$

where σ_m is the mean stress resulting from centripetal loading, σ_{al} is the alternating stress, σ_y is the yield stress, and σ_{en} is the design endurance strength. The factor of safety as a function of rpm (with the same caveats as indicated for Figure 3.7) is shown in Figure 3.8.

Radial Blade Displacements

Displacements in the rotor were calculated with the finite element method as described previously. Blade extension resulting from centripetal and temperature effects was calculated using (3.5), which was derived as follows:

$$\frac{\partial u}{\partial r} = \frac{\sigma_r}{E} + \alpha \Delta T$$

$$d\sigma_r = \rho \omega^2 r dr$$

$$\sigma_r(r) = \rho\omega^2(r_o^2 - r^2)/2$$

$$u = \frac{\sigma_r}{E}dr + \alpha\Delta Tdr$$

$$u = \frac{\rho\omega^2}{6E}(2r_o^3 - 3r_o^2r_i + r_i^3) + (r_o - r_i)\alpha\Delta T \quad (3.5)$$

Structural Damping Measurements

Structural damping for the blades was accomplished through the use of the strain gages mentioned previously. The blades were struck and the ring down was recorded on an oscilloscope. The structural damping measurements on the rotor were made with: 1) all blades constrained but the one being struck (upon which the strain gages were mounted), and 2) with varying numbers of blades free with finally all blades free. The results of these measurements are shown in Figure 3.9 and were used for the estimation of the alternating stress (Figure 3.7). Over the range of initial amplitudes tested, the decrements appear to be relatively independent of initial amplitude. The data shown was idealized as having three decrements over the duration of interest. The decrements used are listed in Table 3.1.

Fig. 3.9. Amplitude* versus number of cycles for rotor blades (with nickel plating)

*Amplitude measured in the critical region of the blade root on the excited blade

Table 3.1
Averaged measured decrements as a function of number of oscillations

j	N_j (cycles)	δ_j
1	12	0.062
2	24	0.031
3	65	0.0136

APPENDIX 4

Nozzle Characterization

The nozzle characterization must begin with determination of the initial conditions entering the nozzle. These conditions were calculated by assuming an isenthalpic flashing process between the measured compressed liquid state (temperature and pressure) delivered by the boiler (see Appendix 7) and the measured pressure of the two-phase mixture entering the nozzle. Thermal and kinetic equilibrium are assumed to exist between phases at the entrance to the nozzle.

Two-phase nozzle flow behavior with slip was calculated by two methods. The first, using 1D2FAZ was the only method with which it was possible to estimate a representative droplet size. The Weber number was varied to obtain a calculated thrust coefficient equal to the experimentally measured thrust coefficient, for the particular nozzle being modeled. It was found that the diameter of the droplet assumed as an initial condition did not alter the calculated results for a given initial condition and Weber number.

The Weber number and thrust coefficient may be defined as follows:

$$We = \frac{\rho_V (V_V - V_L)^2 D}{\sigma} \quad (4.1)$$

$$C_T = \frac{F}{\dot{m} V_{is}} = \frac{R V_V + (1 - R) V_L}{V_{is}} \quad (4.2)$$

Critical Weber numbers typically vary between values of 2 and 20 depending upon the flow conditions and fluids involved. In single component flows, boiling phenomena may prove to be a more stringent criteria for droplet size reduction [13]. Variation of the thrust coefficient and droplet diameter is shown in Figure 4.1 as a function of Weber number for the dynamic blade test nozzle based upon 1D2FAZ calculations. As may be seen, the thrust coefficient is a very weak function of Weber number. Representative droplet diameter determination by correlation of experimentally measured and calculated thrust coefficients based upon Weber number are thus rather insensitive.

The second nozzle modeling technique was used to relate all of the experimentally measured quantities and, unlike the present 1D2FAZ calculations, did not require thermal equilibrium between phases in the throat region of the nozzle. It was assumed that the two-phase mixture entering and exiting the nozzle was in thermal equilibrium. However, since only the entrance and exit planes were referenced, thermal nonequilibrium could occur within the nozzle—hence the name: nonequilibrium model. The model was formulated when it was discovered that the mass flow rate calculated using 1D2FAZ was lower than that measured experimentally. It was hypothesized that thermal nonequilibrium conditions must exist at the throat. Thus, vapor is produced (by evaporation) more rapidly in the 1D2FAZ calculation, where thermal equilibrium between phases is assumed at each point in the flow, than in reality. Thus choking is predicted to occur at a lower flow rate with 1D2FAZ than is measured experimentally. The hypothesis

Fig. 4.1. Variation of thrust coefficient, slip ratio, and droplet diameter with Weber number based upon 1D2FAZ calculations.

can be justified by comparing the thermal time constant for droplet/vapor heat transfer (4.3) with characteristic nozzle transit time.

$$\tau \approx \frac{\rho_L D^2 c_{pL}}{6k_L Nu} \quad (4.3)$$

If the droplet thermal time constant is very small compared to the droplet nozzle transit time (less than .01), one may assume thermal equilibrium between phases throughout the nozzle. If the ratio is of the order of .1, however, thermal equilibrium could not be assumed in regions of high slip, such as the throat—though one could well assume thermal equilibrium between phases at the exit plane of the nozzle. Using representative values for variables in (4.3) and comparing the droplet transit time, one can determine that the maximum droplet diameter for thermal equilibrium to be assumed throughout the nozzle is approximately 2.5 microns, while a droplet diameter of 8 microns would allow thermal equilibrium to be assumed at the exit plane. Based upon droplet size estimates (Table 17), thermal equilibrium does not exist in the throat region of the nozzle. The assumption of thermal equilibrium at the exit plane of the nozzle seems to be justified. Thus, the nonequilibrium model appears to be consistent with droplet thermal time constants and the hypothesis that nonequilibrium occurs in the throat region is credible.

The nonequilibrium model consists of numerically [44] solving the two-phase mixture equations of continuity, momentum, and energy.

(4.4-4.6) with the saturation properties for water, given the experimentally measured values of nozzle thrust coefficient, mass flow rate, and nozzle exit pressure.

$$\dot{m}_1 = \rho_2 A_2 [X_2 V_{V2} + (1 - X_2) V_{L2}] \quad (4.4)$$

$$C_r \dot{m}_1 V_{is} = \rho_2 A_2 [X_2 V_{V2}^2 + (1 - X_2) V_{L2}^2] \quad (4.5)$$

$$\dot{m}_1 \left[\frac{V_1^2}{2} + h_1 \right] =$$

$$\rho_2 A_2 [X_2 V_{V2} \left\{ \frac{V_{V2}^2}{2} + h_V(p_2) \right\} + (1 - X_2) V_{L2} \left\{ \frac{V_{L2}^2}{2} + h_L(p_2) \right\}] \quad (4.6)$$

The effect of the boundary layer on nozzle performance was calculated using a partially developed [32] two-phase boundary layer calculation technique. Presently, the technique does not address the phases individually but can only be used assuming the average mixture properties. This is, of course, consistent with a one-dimensional nozzle flow calculation. The mixture density and pressure profile calculated with 1D2FAZ were used for a flat plate boundary layer analysis. The momentum associated with the boundary layer was less than 1 percent of the nozzle momentum, and therefore boundary layers were neglected.

The nozzle extension and shroud, Figures 6, 32, and 4.2, was used to direct the flow from the nozzle to the rotating blades. Testing [39]

Fig. 4.2. Inside nozzle profile including nozzle extension and shroud.

Fig. 4.3. Oblique shock formation geometry.

of the nozzle established that additional losses were to be expected when the shroud and extension were used (Table 17). It was hypothesized that the nozzle shroud and extension, which caused an abrupt straightening of the flow compared to the divergence (in both directions) of the nozzle, introduced an oblique shock wave (Figure 4.3). The vapor entering the extension region was calculated to have a Mach number, relative to the vapor, greater than 1.5. To investigate the oblique shock hypothesis, both the nonequilibrium and the 1D2FAZ calculations were used to indicate the conditions in the flow entering the shroud. Based upon droplet size estimates, one would not expect significant velocity or temperature change in the liquid phase through the shock. For example, assuming the vapor velocity change was as high as 100 m/s, the droplet velocity gradient may be estimated as only 20 m/s/m according to the relation:

$$\frac{dV_L}{dz} = \frac{3\rho_V(V_V - V_L)^2}{4\rho_L V_L D} C_D \quad (4.7)$$

Thus, the droplet velocity would be expected to change very little through the length of the nozzle extension and shroud if an oblique shock was present. For the purposes of the calculation, the droplet velocity was assumed to be constant through the oblique shock region. The assumption of negligible heat transfer has been previously established and need not be further discussed here.

The equations of continuity, momentum, and energy, for a two-phase oblique shock (4.8-4.11) were solved numerically [44] with saturation

properties. The use of saturation properties assumed thermal equilibrium existed following the shock. This was done for ease of solution. It was then assumed that the mass flow rate ratio remained constant and that the thermal energy associated with any evaporation that occurred went into vapor superheat.

$$\rho_1 [X_1 V_{V1} \sin \theta + (1 - X_1) V_{L1} \sin \theta] = \rho_2 [X_2 V_{V2} \sin(\theta - \delta) + (1 - X_2) V_{L1} \sin \theta] \quad (4.8)$$

$$\rho_1 [X_1 V_{V1}^2 \sin^2 \theta + (1 - X_1) V_{L1}^2 \sin^2 \theta] + p_1 = \rho_2 [X_2 V_{V2}^2 \sin^2(\theta - \delta) + (1 - X_2) V_{L1}^2 \sin^2 \theta] + p_2 \quad (4.9)$$

$$V_{V1} \cos \theta = V_{V2} \cos(\theta - \delta) \quad (4.10)$$

$$\rho_1 [X_1 V_{V1} \sin \theta \left\{ \frac{V_{V1}^2}{2} + h_V(p_1) \right\} + (1 - X_1) V_{L1} \sin \theta \left\{ \frac{V_{L1}^2}{2} + h_L(p_1) \right\}] = \rho_2 [X_2 V_{V2} \sin(\theta - \delta) \left\{ \frac{V_{V2}^2}{2} + h_V(p_2) \right\} + (1 - X_2) V_{L1} \sin \theta \left\{ \frac{V_{L1}^2}{2} + h_L(p_2) \right\}] \quad (4.11)$$

Oblique shock effects were calculated only in the plane of largest

nozzle divergence (11.6 degrees). The 2 degree divergence in the perpendicular plane was neglected. Calculations were made for a variety of deflection angles equal to and less than the nozzle divergence. The resultant velocities were then momentum averaged to obtain the one-dimensional results reported in Table 18.

During dynamic blade testing, measured nozzle thrust coefficients were reduced (Table 17) below those measured previously with the nozzle alone. This was thought to be the result of a radial pressure gradient at the exit of the nozzle which was imposed by the rotating blades. It was noted [39] during nozzle testing that this particular nozzle was sensitive to exit pressure variations. Exit pressures above the optimum pressure were seen to produce unstable flow conditions in which the majority of the exiting fluid was following one wall of the nozzle. It was hypothesized that the additional losses were associated with two mechanisms: 1) oblique shock formation due to flow diversion and separation and/or 2) droplets striking and being slowed by shroud. A combination of the two mechanisms was difficult to model, however, bounds could be estimated if each mechanism was treated independently.

Prediction of the effect of an oblique shock introduced by separation and flow diversion due to radial pressure gradient is not presently possible. It was reasoned that the probable worst result would be the reduction of the vapor velocity so that no slip occurred between phases. This substantially reduces the kinetic energy associated with the vapor which is more efficiently converted to power in the rotor than is the liquid kinetic energy. This assumption also makes

estimation of the nozzle exit condition a straight-forward calculation based only upon (4.2) and the measured thrust coefficient.

In the second case, the nozzle performance with the shroud was calculated using experimental results as previously described. It was then assumed that a fraction of the liquid mass flow rate struck the nozzle wall and was decelerated to negligible velocity. The fraction striking the wall was calculated by assuming that the change in thrust coefficient between the shrouded and unshrouded cases was only a result of this mechanism. The mass flow rate of liquid was thus estimated by using the following relation:

$$\dot{m}_{cn} = \dot{m}_{L2} - \frac{V_{V2}}{V_{L2}} \left[C \frac{V_{is}}{V_{V2}} (\dot{m}_{L2} + \dot{m}_{V2}) - \dot{m}_{L2} \right] \quad (4.12)$$

The liquid leaving the nozzle at low velocity, $\dot{m}_{cn}$, represents an additional loss to the turbine and must also be accounted for there. This is described in Appendix 5.

APPENDIX 5

Turbine Performance Model

The numerical model for the impulse turbine was developed for three reasons: (1) to estimate the performance of the laboratory rotor, (2) to predict the results of increasing admission, and (3) to determine what changes in flow characteristics would lead to the greatest improvements in performance. The rotor model includes the velocity vector equations for the liquid and for the vapor phases, the development of the gross rotor power production equations, and the equations for a control volume within the blade passage written in the relative frame. The losses associated with radial acceleration of droplets adhering to the blade profile (Coriolis pumping losses) are then derived, followed by a description of the losses which do not include the flow passage: disk friction, windage, and partial admission loss, and blade tip leakage.

Blade geometry and related variables are defined in Figure 5.1. The blade geometries for all performance predictions related to the dynamic blade test rotor were specified to simulate the blades and blade spacing of that rotor. The calculations represented in Figure 35, however, assumed an optimum blade spacing derived for vapor turbines [22] based both upon blade width and turning angle. The relationship used may be written as follows:

$$b = \frac{0.4B}{\sin(\pi - 2\beta)} \quad (5.1)$$

Fig. 5.1. Blade geometry and related variables.

The equations for each phase of the two-phase mixture may be written for the entrance and exit of the rotor. The nozzle direction is adjusted so that the vapor flow direction is always tangent to the pressure surface at the entrance to the blades. This is done so that incidence loss need not be accounted for. This was also convenient in determining the best nozzle angle for the rotor, since rotor geometry as fixed long before this model was completed.

Entrance Conditions: Droplets and Vapor

$$V_{V1} \cos \theta_1 = U_1 + V_{RV1} \cos \beta \quad (5.2)$$

$$V_{L1} \cos \theta_1 = U_1 + V_{RL1} \cos \beta_{L1} \quad (5.3)$$

$$V_{V1} \sin \theta_1 = V_{RV1} \sin \beta \quad (5.4)$$

$$V_{L1} \sin \theta_1 = V_{RL1} \sin \beta_{L1} \quad (5.5)$$

Exit Conditions: Droplets Impact Blades

$$V_{L2s}^2 = U_2^2 + V_{rRL2}^2 \quad (5.6)$$

$$V_{RL2s} = V_{rRL2} \quad (5.7)$$

Exit Conditions: Droplets Do Not Impact Blades

$$U_1 - V_{RL2e} \cos \beta_{L2} = V_{L2e} \cos \theta_{L2} \quad (5.8)$$

$$V_{RL2e} \sin \beta_{L2} = V_{L2e} \sin \theta_{L2} \quad (5.9)$$

Exit Conditions: Vapor

$$U_1 - V_{RV2} \cos \beta = V_{V2} \cos \theta_{V2} \quad (5.10)$$

$$V_{RV2} \sin \beta = V_{V2} \sin \theta_{V2} \quad (5.11)$$

One can develop the continuity equation for the two-phase mixture entering and leaving the rotor (5.12).

$$\dot{m}_{V1} + \dot{m}_{L1} = \dot{m}_{V2} + \dot{m}_{L2} \quad (5.12)$$

In order to develop an equation for power output of a two-phase turbine one may proceed in a fashion similar to that used for a single phase turbine. The First Law may be written with reference to two control volumes. Both control volumes lie slightly outside the rotor; one is rotating with the rotor at the rotor speed and the other is fixed in space. So long as the fluid enters and leaves the turbine at the same radius, the absolute and relative frames may be thought of as if the turbine blade pitch line had infinite radius. When fluid radius changes occur between the entrance and the exit, the work per unit mass of fluid experiencing a radius change must be accounted for in the relative frame equations. This work, as a function of radius, is simply the change in kinetic energy per unit mass referenced to the axis of rotation:

$$\Delta KE = \frac{1}{2}(r\omega)^2 = \frac{1}{2}U^2 \quad (5.13)$$

which may be thought of as a potential function.

For the two-phase turbine, it is assumed that the vapor enters and leaves the rotor at the same radius. Liquid droplets, on the other

hand, which are not small enough to pass through the blade passages without impact, adhere to the blade surfaces and are acted upon by centripetal acceleration—imparting radial motion to the liquid. Thus, the liquid portion of the energy equation for the turbine must include the possibility of different entrance and exit radii.

The liquid portion of the mass flow rate leaving the rotor may be divided into two parts: one part which adheres to the blades and experiences radius change, the other part which is associated with droplets small enough so that impact and hence radius change does not occur. The continuity equation (5.12) may be rewritten:

$$\dot{m}_{V1} + \dot{m}_{L1} = \dot{m}_{V2} + \dot{m}_{L2s} + \dot{m}_{L2e} \quad (5.14)$$

The energy equation for the absolute control volume may be written:

$$\begin{aligned} \dot{m}_{V1} \left(h_{V1} + \frac{V_{V1}^2}{2} \right) + \dot{m}_{L1} \left(h_{L1} + \frac{V_{L1}^2}{2} \right) &= \dot{m}_{V2} \left(h_{V2} + \frac{V_{V2}^2}{2} \right) + \dot{m}_{L2e} \left(h_{L2} + \frac{V_{L2e}^2}{2} \right) \\ &+ \dot{m}_{L2s} \left(h_{L2} + \frac{V_{L2s}^2}{2} \right) + \dot{W} \end{aligned} \quad (5.15)$$

the energy equation for the relative control volume, including potential functions for change for both the vapor and the liquid, may be written:

$$\begin{aligned} \dot{m}_{V1} \left(h_{V1} + \frac{V_{RV1}^2}{2} - \frac{U_1^2}{2} \right) + \dot{m}_{L1} \left(h_{L1} + \frac{V_{RL1}^2}{2} - \frac{U_1^2}{2} \right) &= \dot{m}_{V2} \left(h_{V2} + \frac{V_{RV2}^2}{2} - \frac{U_1^2}{2} \right) \\ &+ \dot{m}_{L2e} \left(h_{L2} + \frac{V_{RL2e}^2}{2} - \frac{U_1^2}{2} \right) + \dot{m}_{L2s} \left(h_{L2} + \frac{V_{RL2s}^2}{2} - \frac{U_1^2}{2} \right) \end{aligned} \quad (5.16)$$

Applying continuity (5.14) and rearranging terms, (5.16) may be rewritten:

$$\begin{aligned} \dot{m}_{V1} \left(h_{V1} + \frac{V_{RV1}^2}{2} \right) + \dot{m}_{L1} \left(h_{L1} + \frac{V_{RL1}^2}{2} \right) = \dot{m}_{V2} \left(h_{V2} + \frac{V_{RV2}^2}{2} \right) \\ + \dot{m}_{L2e} \left(h_{L2} + \frac{V_{RL2e}^2}{2} \right) + \dot{m}_{L2s} \left(h_{L2} + \frac{V_{RL2s}^2}{2} + \frac{U_1^2}{2} - \frac{U_2^2}{2} \right) \end{aligned} \quad (5.17)$$

The energy equations for the absolute and relative frames (5.15) and (5.17) may now be subtracted to yield the turbine energy equation in terms of velocities only:

$$\begin{aligned} \dot{m}_{V1} (V_{V1}^2 - V_{RV1}^2) + \dot{m}_{L1} (V_{L1}^2 - V_{RL1}^2) = \dot{m}_{V2} (V_{V2}^2 - V_{RV2}^2) + \dot{m}_{L2e} (V_{L2e}^2 - V_{RL2e}^2) \\ + \dot{m}_{L2s} (V_{L2s}^2 - V_{RL2s}^2 + U_2^2 - U_1^2) + 2\dot{W} \end{aligned} \quad (5.18)$$

Using the relations (5.2) through (5.11), and assuming that liquid droplets impacting the blade are reduced to negligible velocities in the direction of vapor flow, (5.18) may be rewritten:

$$\begin{aligned} \dot{m}_{V1} (2U_1 V_{V1} \cos \theta_1 - U_1^2) + \dot{m}_{L1} (2U_1 V_{V1} \cos \theta_1 - U_1^2) = \dot{m}_{V2} (U_1^2 - 2U_1 V_{RV2} \cos \beta) \\ + \dot{m}_{L2e} (U_1^2 - 2U_1 V_{RL2e} \cos \beta_{L2}) + \dot{m}_{L2s} (2U_2^2 - U_1^2) + 2\dot{W} \end{aligned} \quad (5.19)$$

Note that (5.19) is independent of the radial velocity of the liquid on

the blade relative to the blade surface, V_{RL2} . It is convenient to subtract and add the term: $\dot{m}_{L2s} U_1^2$ to (5.19) so that the resultant equation is:

$$\begin{aligned} \dot{W} = & \frac{1}{2} [\dot{m}_{V1} (2U_1 V_{V1} \cos \theta_1 - U_1^2) + \dot{m}_{L1} (2U_1 V_{V1} \cos \theta_1 - U_1^2) + \\ & \dot{m}_{V2} (2U_1 V_{RV2} \cos \beta - U_1^2) + \dot{m}_{L2e} (2U_1 V_{RL2e} \cos \beta_{L2}) - \dot{m}_{L2s} U_1^2] \\ & - \dot{m}_{L2s} (U_2^2 - U_1^2) \end{aligned} \quad (5.20)$$

The term (5.20) which is multiplied by one-half is the expression one would obtain if no radius change occurred in either the liquid or the vapor, i.e. a pitchline analysis. The second term is associated with Coriolis work and is the work required to move (or pump) the liquid from the entering radius to a larger exiting radius. To show this, consider an element of fluid in a fluid stream moving radially with velocity V_r . The force, dF , on the differential mass dm may be derived:

$$\begin{aligned} dF &= 2\omega V_r dm \\ dm &= 2\rho A dr \\ \dot{m} &= \rho A V_r \\ dF &= 2\dot{m} \omega dr \end{aligned} \quad (5.21)$$

The differential torque dM exerted by the element on the rotor blade is:

$$dM = r dF \quad (5.22)$$

Finally, the work rate may be written:

$$\dot{W}_C = \int_{r_1}^{r_2} 2\dot{m}\omega^2 r dr \quad (5.23)$$

$$\dot{W}_C = \dot{m}_{L2s} (U_2^2 - U_1^2) \quad (5.24)$$

which results in the second term of equation (5.20).

The net power output may thus be thought of as the power determined with a pitchline analysis less the Coriolis pumping power, $\dot{W}_C$, and the disk friction and windage losses, $\dot{W}_D$:

$$\begin{aligned} P = & \frac{1}{2} [\dot{m}_{V1} (2U_1 V_{V1} \cos\theta_1 - U_1^2) + \dot{m}_{L1} (2U_1 V_{V1} S \cos\theta_1 - U_1^2) + \\ & \dot{m}_{V2} (2U_1 V_{RV2} \cos\beta - U_1^2) + \dot{m}_{L2e} (2U_1 V_{RL2e} \cos\beta_{L2}) - \dot{m}_{L2s} U_1^2] \\ & - \dot{W}_C - \dot{W}_D \end{aligned} \quad (5.25)$$

Both the Coriolis pumping and the disk friction and windage losses will be dealt with in more detail later.

In order to complete the solution, droplet trajectories must be calculated to determine the amount of liquid striking the blades. A blade flow energy equation must be developed to correct the vapor velocity for the effect of droplet coupling to the vapor, and some assumptions must be made with respect to vapor flow loss mechanisms, droplet evaporation, and the exit slip ratio when droplets are small enough to pass through the blade passages without impact.

Droplet trajectories were calculated in the relative frame at the pitchline (assuming a two-dimensional analysis only) by solving the droplet momentum equations (5.26) and (5.27) for the tangential and radial directions. The blade passages are assumed to be comprised of concentric circular arcs. The vapor velocity profile is assumed to be uniform across the blade channel, always tangent to the contiguous blade surface, and constant throughout the length of the blade channel during the trajectory calculation.

$$\frac{1}{2}\rho_V C_D \pi \frac{D^2}{4} (V_{\theta V} - r\dot{\theta})V_{rel} = \rho_L \pi \frac{D^3}{6} (r\ddot{\theta} + 2\dot{\theta}\dot{r}) \quad (5.26)$$

$$\frac{1}{2}\rho_V C_D \pi \frac{D^2}{4} (V_{rV} - \dot{r})V_{rel} - \pi \frac{D^3}{6} \frac{V_{\theta V}^2}{r} = \rho_L \pi \frac{D^3}{6} (\ddot{r} - r\dot{\theta}^2) \quad (5.27)$$

where

$$V_{rel} = \sqrt{(V_{\theta V} - r\dot{\theta})^2 + (V_{rV} - \dot{r})^2}$$

The term for the radial pressure gradient (for vapor alone) is included in the radial momentum equation (5.27) though its effect is small. This pressure gradient would be somewhat larger for a completely coupled solution. Droplet evaporation is neglected during the droplet trajectory calculation. Evaporation would have a minimal effect on the diameter, even through large pressure changes. Since this is an impulse turbine calculation, only minor pressure variations would be expected. Because of the complexity of the equations, a numerical solution [45] was used.

It was expected that droplet trajectories would be relatively independent of the blade passage vapor velocity profile. To verify this, droplet trajectories were calculated for a uniform vapor velocity distribution and for a free vortex vapor velocity distribution. These trajectories were calculated for a range of droplet diameters and initial vapor velocities representing all of the conditions encountered herein. The results showed that for a given droplet diameter and initial condition, the location at which the droplet left the boundary of the blade channel differed by less than 3 percent for the two vapor velocity distributions. Without solving the fully coupled equations, the actual vapor velocity profile cannot be determined, the simplest profile is thus assumed since the more complex profile is neither required nor justified.

The velocities and displacements for the trajectories calculated from the momentum equations were approximated with quadratic and cubic functions. These functions were numerically overlayed on the blade

profile so that impact locations and velocities could be calculated, using a root finder [46]. Thus, it was possible to determine the fraction of liquid passing freely through the blade passage and the remainder which impacted the blades. Average velocities of droplets impacting the blades were also calculated. These velocities will be described later in the development of the blade energy equation source terms. The exit angle of the droplets, β_{L2} , (5.10) which passed through the blades without impact was calculated by mass averaging the exit angle of all such trajectories.

The trajectory calculations also provided the means to determine the space averaged droplet impact pressures for the static cascade pressure profile correlations. The space averaged droplet impact pressure may be defined:

$$\Delta p_{IM} = \rho_L (1 - \epsilon) V_L^2 \quad (5.28)$$

A liquid mass flow rate is associated with each trajectory. The spacing between trajectories is known at the blade entrance and can be calculated at the location of droplet/blade impact. Thus, all of the geometry indicated in Figure 5.2 is known or may be calculated using the trajectory and the known blade geometry.

For a steady state solution, the droplet number rate is constant along a trajectory. This leads to the following expressions:

$$NV_L A = \text{constant}$$

Fig. 5.2. Geometry for static blade space averaged impact pressure calculation.

$$(1 - \epsilon_1)V_{L1}A_1 = (1 - \epsilon_2)V_{L2}A_2 \quad (5.29)$$

When thermal equilibrium is assumed to exist between phases, the void fraction and mass fraction may be related:

$$\rho(1 - X) = \rho_L(1 - \epsilon) \quad (5.30)$$

By combining the geometric relations (Figure 5.2) and equations (5.28) through (5.30), the expression for the space averaged droplet impact pressure results:

$$\Delta p_{IM} = \frac{\rho(1 - X_1)V_{L1}V_{L2}\Delta b \sin\theta}{r_p(\Delta\phi)} \quad (5.31)$$

The velocities of the liquid and vapor at the exit of the blades in the relative frame remain to be determined. One may consider a control volume, shown in Figure 5.3, which is so positioned as to be inside any liquid or vapor boundary layers associated with the blade surfaces. Again, the pressure at the entrance and exit of the blades is assumed to be equal, which is consistent with a pure impulse machine. The continuity equation (5.14) may again be modified to include evaporation of liquid within the blade passage.

$$\dot{m}_{V1} + \dot{m}_{L1} = \dot{m}_{V2} + \dot{m}_{L2e} + S_L - S_{Vs} - S_{Vh} \quad (5.32)$$

The source term, S_L , represents the liquid leaving the control volume

Fig. 5.3. Control volume relationships for relative reference frame.

near the blade surface. The two vapor source terms, S_{Vs} and S_{Vh} , are the result of liquid evaporation which can be expected to occur in the blade passage. These effects were included for completeness. Two phenomena can provide conversion of liquid to vapor: 1) droplets striking the blade can be assumed to contribute their kinetic energy to isothermal evaporation as indicated in (5.33):

$$S_{Vs} = \sum \dot{m}_j [1 - e^{-\{V_{RL2Bj}^2 / 2(h_{V2} - h_{L1})\}}] \quad (5.33)$$

where

$$\frac{d\dot{m}_j}{\dot{m}_j} = \frac{V_{RL2Bj} dV_{RL2Bj}}{h_{V2} - h_{L1}} \quad (5.34)$$

and where $\dot{m}_j$ is the mass flow rate associated with the j th trajectory, and 2) the enthalpy increase in the vapor due to friction losses (5.35):

$$S_{Vh} = \dot{m}_{V1} \xi (V_{RV1} \lambda_V)^2 / (h_{V2} - h_{L1}) \quad (5.35)$$

The assumption of isothermal evaporation of droplets to maintain thermal equilibrium allows the production of more vapor than would occur in a real (nonisentropic) impact, however evaporation is a small part of the calculation and this provides a reasonable approximation. The energy associated with heating the vapor can either be assumed to

evaporate liquid so that liquid and vapor remain saturated, as in (5.35), or to cause the vapor leaving the blades to be slightly superheated. Friction heating of the vapor is assumed to effect only the continuum phase and is calculated based on vapor turbine correlations.

The vapor velocity coefficient (5.36) for the blades was estimated, as indicated in the text, based upon correlations of static cascade results with the turbine performance model.

$$\lambda_V = V_{RV2}/V_{RV1} \quad (5.36)$$

The enthalpy-related loss coefficient, ξ , (5.37) may be defined in terms of the velocity coefficient:

$$\xi = \frac{1 - \lambda_V^2}{\lambda_V^2} \quad (5.37)$$

Another option was exercised in developing the results shown in Figure 35. In this case, a vapor turbine correlation [22] of the form (5.38) was used.

$$\xi = \left[\frac{10^5 \{H + b \cos(\pi/2 - \beta)\} \mu_V}{2Hb \cos(\pi/2 - \beta) \rho_V V_{RV1}} \right]^{0.25} \cdot \left[\{1 + f(\phi)\} \{0.975 + 0.075 \frac{B}{H}\} - 1 \right] \quad (5.38)$$

where $f(\phi)$ was a cubic spline fit to the loss coefficient curve in [22].

If a truly coupled and discretized calculation had been used, a moment of momentum equation could be written for the two-phase mixture between the blades. However, this cannot be done here since the fluid states at stations between the blade entrance and exit required for such an equation to be applied are not present in this approach. The energy equation for the control volume can be developed, however, and from this, the vapor velocity can be found. The mixture energy equation for the control volume is written:

$$\begin{aligned} \dot{m}_{V1} \left(h_{V1} + \frac{V_{RV1}^2}{2} \right) + \dot{m}_{L1} \left(h_{L1} + \frac{V_{RL1}^2}{2} \right) = \dot{m}_{V2} \left(h_{V2} + \frac{V_{RV2}^2}{2} \right) + \\ \dot{m}_{L2e} \left(h_{L1} + \frac{V_{RL2e}^2}{2} \right) + S_L \left(h_{L1} + \frac{\bar{V}_{RL2B}^2}{2} \right) - S_{Vs} \left(h_{V2} + \frac{\bar{V}_{RV2B}^2}{2} \right) - S_{Vh} h_{V2} \quad (5.39) \end{aligned}$$

where

$$\bar{V}_{RL2B} = \frac{\sum V_{RL2Bj} \dot{m}_j}{\sum \dot{m}_j} \quad (5.40)$$

$$\begin{aligned} V_{RV2Bj} &= \frac{\int V_{RV2Bj} d\dot{m}_j}{\int d\dot{m}_j} \\ \bar{V}_{RV2B} &= \frac{\sum V_{RV2Bj} S_{Vs_j}}{\sum S_{Vs_j}} \quad (5.41) \end{aligned}$$

The velocities $\bar{V}_{RL2B}$ and $\bar{V}_{RV2B}$ are momentum averaged velocities of all droplets leaving the control volume (5.40) and all vapor entering the control volume (5.41) along the blade surface, respectively. Equation (5.41) is derived by assuming that vapor is emitted according to (5.33) while the droplet is sliding to rest on the blade. These equations are

written in terms of the liquid mass flow rate represented by the j th droplet trajectory. If the liquid is assumed to evaporate in order to maintain temperature equilibrium between phases, the vapor enthalpies entering and leaving the rotor are equal—otherwise, the vapor leaving the blades is slightly superheated (5.42).

$$h_{V2} = h_{V1}(1 + \xi) \quad (5.42)$$

It has been found for all combinations of blade geometries and droplet sizes of interest that if the droplets are small, they pass through the blade passages without impact. Such droplets may be assumed to have achieved the same magnitude of velocity as the vapor at the exit of the blades. For larger droplets, blade impact eliminates the need to calculate the liquid velocity exiting the blades. Equation (5.39) can thus be simplified as shown in (5.43)

$$V_{RV2} = V_{RL2e} = \sqrt{\frac{2\dot{m}_{V1}(h_{V1} - h_{V2}) + \dot{m}_{V1}V_{VR1}^2 + S_{Vs}\bar{V}_{RV2B}^2 - S_L\bar{V}_{RL2B}^2}{\dot{m}_{V2} + \dot{m}_{L2e}}} \quad (5.43)$$

Thus, with this assumption, all of the terms of (5.25), other than the Coriolis pumping power and the disk friction and windage loss may be determined. A more detailed derivation of the Coriolis pumping loss is next derived.

Coriolis Pumping Including Liquid Film Calculations

Before further describing the loss associated with radial removal of liquid impacting the blades (Coriolis pumping loss), it is appropriate to consider the forces acting on an element of a liquid layer or rivulet on the blade surface. It is assumed that most of the liquid would initially reside on the pressure surface of the blade, a short time after droplet impact. In the direction of flow (x-direction, see Figure 5.4) the forces include: the droplet momentum source (due to droplet impact) (5.44), the vapor drag (5.45), the liquid/wall shear force for Couette flow (5.46), and if radial motion occurs, the Coriolis force (5.47).

$$\tau_L = \rho_L (1 - \epsilon) V_{RL2B}^2 \cos \gamma \quad (5.44)$$

$$\tau_V = C_F \rho_V \frac{V_{RV1}^2}{2} \quad (5.45)$$

$$\tau_{Lx} = -\mu \frac{V_x}{\delta} \quad (5.46)$$

$$\tau_C = -2\rho_L V_y \frac{U}{r} \delta \cos \psi \quad (5.47)$$

In the y-direction, the liquid/wall shear force (5.48) resists the centripetal force (5.49).

$$\tau_{Ly} = -\mu \frac{\bar{V}_y}{\delta} \quad (5.48)$$

$$\tau_r = \rho_L \frac{U^2}{r} \delta \quad (5.49)$$

Fig. 5.4. Details of the calculational grid for the liquid layer.

Normal to the blade surface, the forces acting on the liquid layer are the vapor pressure (5.50), the Coriolis force (5.51), and the normal component of the momentum source.

$$\sigma_p = -\rho_V V_{RVI}^2 \ln(r_p/r_s) \quad (5.50)$$

$$\sigma_C = 2\rho_L \bar{V}_y \frac{U}{r} \delta \sin\psi \quad (5.51)$$

The incidence angle, γ , for the droplet wall momentum source will be assumed to be zero for the remainder of the discussion.

The surface tension forces, coupled with surface instabilities tend to make the liquid layer form rivulets. In addition, when the forces causing the liquid layer to adhere to the pressure surface are overcome, large liquid droplets can be removed from the pressure surface by Coriolis acceleration. This leads to vapor flow restriction and possible entrainment of liquid into the passing vapor.

As may be seen, the centripetal force, the Coriolis force, and the liquid/wall shear force are dependent upon the liquid layer thickness. For this reason, it is inconclusive to compare force magnitudes in the x- and y- directions without an approximation of the liquid layer thickness. The calculation of rivulet formation, surface instabilities, and liquid entrainment will not be attempted. However, one can do a reasonably straight-forward two-dimensional analysis of a continuous liquid layer on the pressure surface to provide insight.

For this calculation, full admission, i.e. continuous flow through the

blade passages is assumed. The continuity equation (5.52) and the momentum equations in the x- and y-directions (5.53) and (5.54), respectively, were written for a small element of dimensions Δx by Δy on the curved pressure surface of the blade. These equations were integrated in the liquid layer thickness direction by assuming a velocity distribution. Two velocity distributions which bound the possible conditions were included: the first assumes Couette flow, while the second assumes that the liquid/wall shear force is zero with the velocity distribution being uniform. These velocity profiles were selected to bound the possible liquid layer conditions so that minimum and maximum liquid layer thickness distributions would result.

$$\begin{aligned}
 & - \int_0^{\bar{\delta}_{x1}} \rho_L V(z)_{x1} \Delta y dz + \int_0^{\bar{\delta}_{x2}} \rho_L V(z)_{x2} \Delta y dz - \int_0^{\bar{\delta}_{y1}} \rho_L V(z)_{y1} \Delta x dz \\
 & + \int_0^{\bar{\delta}_{y2}} \rho_L V(z)_{y2} \Delta x dz = s \Delta x \Delta y
 \end{aligned} \tag{5.52}$$

$$- \int_0^{\bar{\delta}_{x1}} \rho_L V(z)_{x1}^2 \Delta y dz + \int_0^{\bar{\delta}_{x2}} \rho_L V(z)_{x2}^2 \Delta y dz = \Sigma \tau_x \tag{5.53}$$

$$- \int_0^{\bar{\delta}_{x2}} \rho_L V(z)_{y1}^2 \Delta x dz + \int_0^{\bar{\delta}_{x2}} \rho_L V(z)_{y2}^2 \Delta x dz = \Sigma \tau_y \tag{5.54}$$

where (from Figure 5.4)

$$\begin{aligned}
 \bar{\delta}_{x1} &= \frac{\delta_1 + \delta_2}{2} \\
 \bar{\delta}_{x2} &= \frac{\delta_3 + \delta_4}{2}
 \end{aligned}$$

$$\bar{\delta}_{y1} = \frac{\delta_1 + \delta_4}{2}$$

$$\bar{\delta}_{y2} = \frac{\delta_2 + \delta_3}{2}$$

$$\bar{\delta} = \frac{\delta_1 + \delta_2 + \delta_3 + \delta_4}{4}$$

and

$$\bar{V}_y = \frac{V_{y1} + V_{y2}}{2}$$

$$\tau_L = s V_{RL2B}$$

Having integrated in the thickness direction, the continuity equation (5.52) may be rewritten (5.55), where the velocities are now the velocities associated with the liquid/vapor interface. Similarly, the momentum equations (5.53) and (5.54) may be rewritten as (5.56) and (5.57) with the various forces included. Pressure gradient in the flow direction was neglected because it would be expected to be small in impulse blading. The constants CON2 and CON3 are 2 and 3, respectively, for Couette flow and 1 for the uniform velocity distribution.

$$\frac{\rho_L}{\text{CON2}}[(V_{x2}\bar{\delta}_{x2} - V_{x1}\bar{\delta}_{x1})y + (V_{y2}\bar{\delta}_{y2} - V_{y1}\bar{\delta}_{y1})\Delta x] = s\Delta x\Delta y \quad (5.55)$$

$$\begin{aligned} \frac{\rho_L}{\text{CON3}}[(V_{x2}^2\bar{\delta}_{x2} - V_{x1}^2\bar{\delta}_{x1})\Delta y - (\tau_V + \tau_{Lx})\Delta x\Delta y - \frac{\tau_C}{\text{CON2}}\Delta x\Delta y = \\ s V_{RL2B}\Delta x\Delta y \end{aligned} \quad (5.56)$$

$$\frac{\rho_L}{\text{CON3}}[(V_{y2}^2\bar{\delta}_{y2} - V_{y1}^2\bar{\delta}_{y1})\Delta x - (\tau_{Ly} + \tau_C)\Delta x\Delta y] = 0 \quad (5.57)$$

In order to conveniently integrate these equations in the x-y surface, it was necessary to consider only elements with a momentum source, i.e. elements in which droplet impact occurred. The droplet momentum source term was large compared to all other terms in the momentum equation. Since the integration scheme required that convection be both in the +x and +y directions, this could only be achieved (with Couette flow) when the momentum source term occurred over the total blade surface. The equations (5.55), (5.56), and (5.57) were solved with an implicit numerical scheme, marching from $x=0$, $y=0$ first in the x-direction and then in the y-direction—using a root finder [46] to solve the final equation in terms of the other two.

The results, for blade geometry and velocities representative of the rotor tested (but at peak full admission operating conditions) revealed the following information. 1) The droplet momentum source and the liquid/wall shear (Couette) terms are dominant because the liquid layer is quite thin (.3 to 1.5 microns). 2) The assumption of Couette flow is reasonable, based on the calculated layer thicknesses. It may be argued that the y-velocity profile, which is not acted upon by the droplet momentum source, would better be modeled with a velocity profile between the uniform and the linear Couette profile. This would further thin the layer, making the flow tend to have a larger y-velocity component than would be calculated with the Couette approximation. The result would thus be to increase the Coriolis loss compared to the present analysis. 3) It is unlikely for liquid to exit

the blade in the x-direction unless droplets continuously impinge the blade over its entire surface. 4) If droplet impingement does occur over the whole blade surface, Coriolis losses can be significantly reduced because the liquid tends to exit the blade in the x-direction with little radial change in transit across the blade. 5) Vapor shear force at the liquid/vapor interface is not significant in altering the liquid layer flow field in the x-direction. 6) Coriolis force (5.51) becomes larger than the vapor centripetal pressure forces (5.50) normal to the blade surface for over 50 percent of the blade surface area, leading to probable liquid entrainment in the passing vapor; the normal component of the momentum source was neglected.

Since these results do not include surface tension forces and possible instabilities at the liquid/vapor interface, conclusions drawn from the previous results include some uncertainty. Based upon these results it is concluded that for the partial admission expander, where the flow forces including the droplet momentum source term are not continuous (for the single-nozzle configuration tested, the flow forces acted only 4 percent of the time on a given blade), all of the liquid striking the blade exits radially. For the full-admission machine, if the droplets are small enough to provide a momentum source over the whole blade pressure surface, a large fraction of the liquid (60 to 90 percent) can leave the blade in the x-direction with small radius change in transit across the blade. If, however, the droplets are larger, so that impact occurs only over a portion of the pressure surface, the majority of the liquid impacting the blade will exit radially. It appears probable that with rivulet formation, the previous conclusions hold. This does

not address the entrainment problem. Certainly, a fraction of the entrained liquid might be expected to pass through the blade passage without further impact. Some of the droplets would reattach themselves to the suction surface. The Weber break-up effectiveness would be considerably reduced, compared to the throat of the nozzle, leading one to conclude that most entrained liquid would find its way to the suction surfaces because of large droplet size. Those droplets reaching the suction surface would be expected to be removed radially, based on the previous discussion.

The upper bound on the Coriolis loss may be derived by assuming that all droplets striking the blades are removed radially. Droplets are assumed to be uniformly distributed over the exit of the nozzle. Hence, the droplets may be assumed to be evenly distributed along the blade height in the region of the blade supplied with fluid by the nozzle. The mass flow rate is thus a function of radius (5.58), where r_i is the radius of the blade associated with the bottom of the nozzle (Figure 6), and r_f is the radius of the blade associated with the top of the nozzle.

$$\dot{m}(r) = \dot{m}_{L2s} \frac{r - r_i}{r_f - r_i} \quad (5.58)$$

The expression for mass flow rate as a function of radius may be used in (5.23) to obtain (5.59).

$$\dot{W}_{Clf} = \omega^2 \dot{m}_{L2s} \left(\frac{2r_f^2 - r_f r_i - r_i^2}{3} \right) \quad (5.59)$$

Because of the lap (blade height greater than the nozzle height), an addition term must be added to the previous expression. This term is the result of the total mass flow rate moving from radius r_f to r_o :

$$\dot{W}_{Cfo} = \omega^2 \dot{m}_{L2s} (r_o^2 - r_f^2) \quad (5.60)$$

Finally, the expression for the total power losses associated with Coriolis pumping is found by adding (5.59) and (5.60).

$$\dot{W}_C = \dot{W}_{Clf} + \dot{W}_{Cfo} = \omega^2 \dot{m}_{L2s} \left[\left(\frac{2r_f^2 - r_f r_i - r_i^2}{3} \right) + (r_o^2 - r_f^2) \right] \quad (5.61)$$

Losses Associated With Low Velocity Liquid Exiting the Nozzle

The effect of liquid exiting the nozzle at low velocity is two-fold. The power loss associated with radial liquid removal from the blades, and the liquid must be accelerated to the rotor velocity, both of which represent additional losses to the rotor. If inelastic collision between the low velocity liquid and the blade is assumed, the additional loss to the rotor is twice the kinetic energy change, since the liquid internal energy is raised by an amount equal to the kinetic energy change. The expression for this loss (5.62) is derived as follows:

$$\dot{W}_P = \int_{r_i}^{r_f} (\omega^2 r^2 - V_I^2) d\dot{m}$$

$$d\dot{m} = \frac{\dot{m}}{r_f - r_i} dr$$

$$\dot{W}_P = \dot{m}_{cn} \left[\frac{\omega^2}{3} (r_f^2 + r_f r_i + r_i^2) - V_I^2 \right] \quad (5.62)$$

Disk Friction and Windage, Partial Admission, and Tip Clearance Losses

The model used for initial predictions of disk friction and windage was based upon empirical measurements of bladed and unbladed disks in housings [47]. Because the annulus adjacent to the exit side of the blades was open in the design under consideration, it would be expected that the enclosed housing model would underpredict losses, as was later determined experimentally.

The model used to correct the measured disk friction and windage loss of the rotor to a full admission condition was also based upon [47]. It was assumed that the active portion of the blades did not contribute to the disk friction loss. The data in [47] included information relating blade aspect ratio (H/B) to rotor moment coefficient. The disk friction and windage power loss may be approximated as shown in (5.63).

$$\dot{W}_D = \omega (M_1 - M_2 + M_3 + M_4 - M_5) \quad (5.63)$$

where M_1 is the moment due to inactive blades and the disk, M_2 is the moment due to the inactive disk, M_3 is the moment due to the disk alone, M_4 is the moment due to the bladed disk with blade aspect ratio equal to the full blade height, and M_5 is the moment due to the bladed

disk with the blade aspect ratio equal to the blade height corresponding to the nozzle height. M_1 and M_2 were assumed to be linear functions of the admission angle, θ . The form of the moment coefficient used to fit the empirical data is shown in (5.64), where G and ζ were empirically determined. The expression for G (5.66) is for the specific geometry of the rotor and rotor clearances tested.

$$C_M = G R_E^{-\zeta} \quad (5.64)$$

$$R_E = \frac{\rho \omega r_o^2}{\mu} \quad (5.65)$$

$$G = 0.0555(1 + H/B) \quad (5.66)$$

$$C_M = \frac{2M}{\rho \omega^2 r_o^5} \quad (5.67)$$

Using (5.64) and the definition of the moment coefficient (5.67), one may rewrite (5.63):

$$\begin{aligned} \dot{W}_D = 0.0278 C \left[\left(\left(1 + \frac{r_f - r_i}{B} \right) r_f^{5-2\zeta} - r_i^{5-2\zeta} \right) \left(\frac{2\pi - \theta}{2\pi} \right) + r_i^{5-2\zeta} \right. \\ \left. + \left(1 + \frac{r_o - r_f}{B} \right) r_o^{5-2\zeta} - \left(1 + \frac{r_f - r_i}{B} \right) r_f^{5-2\zeta} \right] \rho^{1-\zeta} \mu \omega^{3-\zeta} \end{aligned} \quad (5.68)$$

where C and ζ were 1 and 0.15 prior to disk friction and windage experiments, and were later modified in accordance with the moment coefficients measured in water vapor as indicated in Figure 29.

The validity of superposition may be questioned here. However, since this is the only detailed available data concerning bladed disks in housings, the method was used. Since this model was normalized with measured data for the rotor with zero admission, the most serious questions of validity would be concerned with the full admission predictions. Under these circumstances, assuming all blades are active, a large portion of the losses contributed by blade pumping disappears. Also, the relative fraction of the disk friction and windage losses is much smaller for a full admission machine than for a partial admission machine. Thus, from a pragmatic viewpoint, it is most important to properly model the partial admission case, as has been done here.

The sector loss occurs in partial admission machines only. It is the result of the necessity of accelerating stagnant fluid between the blades each time the nozzles are encountered. The velocity coefficient (5.69) was applied to the relative vapor exit velocity and is based on vapor turbine correlations found in [48].

$$\lambda_{pa} = \left(1 - \frac{b}{3r_m \theta}\right) \quad (5.69)$$

Though leakage of fluid past the blades in the annular region between the blade tips and the rotor housing is far less important in an impulse turbine than in a reaction design, it can contribute a large enough fraction of the total loss so that it should not be neglected. The blade tip leakage loss was accounted for as suggested in [48], which was based on vapor turbine correlations. A fraction, based upon

the rotor and rotor clearance tested (0.92) was multiplied by the vapor produced portion of the power output.

APPENDIX 6

Two-Phase Shock Demonstration

1D2FAZ calculations will herein be used to demonstrate that shocks can form at velocities below the vapor sonic velocity, but above the two-phase mixture sonic velocity. As the momentum coupling between the liquid phase and the vapor phase improves, the two-phase mixture behaves more and more like a dense gas continuum. Momentum coupling increases as the droplet diameter becomes smaller. The demonstration calculations will indicate, for a particular set of fluid conditions, the size of droplets required for dense gas behavior to effect shock structure. The intent of this discussion is not to be a rigorous treatise, but rather to be an introduction to the phenomena described.

A simple nozzle is used for the calculation. The ideal exit velocity of the nozzle is approximately Mach 1.4 relative to the homogeneous mixture choking velocity, and Mach .6 relative to the vapor sonic velocity. A shock is induced in the nozzle by specifying a subsonic exit boundary condition [7], which requires specification of the vapor velocity. A normal shock can thus form, provided the Rankine-Hugoniot conditions for a normal shock have been met or exceeded. As will be shown, the droplet size strongly determines whether the resultant flow disturbance is a sharp discontinuity or a more gradual flow adjustment.

Figure 6.1 shows the nozzle profile and resultant pressure profiles for the liquid and vapor phases for three different droplet sizes. The

Fig. 6.1. Pressure profiles as a function of distance along the nozzle.

vapor velocity at the exit was the same for all three calculations. For the 1 micron diameter droplet, a very sharp discontinuity in velocity is seen at the exit, indicating shock-like behavior. As the droplet diameter was increased to 2 microns and finally to 5 microns, the flow disturbance becomes increasingly gradual.

In Figure 6.2 the nozzle profile is again shown with the resultant velocity profiles. As with the pressure profiles, the discontinuity becomes increasingly gradual as droplet size is increased.

This demonstration does not allow direct comparison of shock phenomena with exactly the same initial conditions entering the shock. Calculationally, in order for the shock to achieve a stable location in the steady flow field, the duct must have some divergence. Because the shock and relaxation region after the shock is generally wide enough in two-phase flow that area change in the diverging duct effects the result, direct comparison calculations are difficult to show.

Fig. 6.2. Velocity profiles as a function of distance along the nozzle.

APPENDIX 7

Description of Supporting Experimental Apparatus and Instrumentation

The dynamic blade and static cascade apparatuses have been described in the text. The purpose of this section is to describe the additional specialized equipment used to obtain the reported experimental results.

The boiler and condenser are described in detail in [18]. The boiler supplies hot water at high pressure. Isenthalpic expansion of the hot water through an adjustable valve, Figures 7.1a and 7.1b, produces a two-phase mixture of steam and water. The desired enthalpy is achieved by adjusting the boiler temperature and pressure. Pressure drop is then adjusted by use of the valve to obtain the desired quality. A shell and tube condenser is used to achieve subatmospheric exhaust conditions. The boiler and condenser were operated in a closed loop mode. Treated boiler water, which behaves sufficiently like pure water that standard equation of state relations are a good representation, was used as the working fluid.

A unique hydraulic dynamometer/drive system was designed [38] for both power performance measurements and determination of disk friction and windage. This consisted of a hydraulic pump/motor which could act as a brake (solid lines), Figure 7.2, or as a motor (dashed lines). In the pump mode, the energy being absorbed at the pump by the fluid was removed in a small heat exchanger. A diesel engine/pump unit provided hydraulic power for disk friction and windage determination in the motor mode. The complete assembly of the hydraulic dynamometer is

Fig. 7.1a. Thermodynamic process description for test.

Fig. 7.1b. Test loop.

Fig. 7.2. Hydraulic dynamometer/drive system.

shown in Figure 7.3. This unit was located below the rotor assembly during operation.

Torque was measured directly at the output shaft of the rotor with a commercial in-line 339 Nm capacity strain gaged torque transducer using coupling transformers for transfer of the electrical power and signal between the stationary and rotating frames. A toothed wheel and proximity transducer were used with a digital counter to measure angular velocity. The range of this device was 0 to 20,000 rpm. Most measurements were made at torque levels less than one-third of the range of the torque sensor and less than one-fifth the range of the angular velocity measuring system. Accuracy could have been improved by better matching of the maximum ranges of the measurement devices to the measurement range. Calibration of the angular velocity measurement device indicated that it was accurate, over the measurement range of interest, to within less than plus or minus 1 percent of the measured value. Dynamic calibration (during rotation) of the torque sensor was not accomplished. Static calibration indicated that it was accurate to within less than plus or minus 1 percent of the measured value, with an upper limit of 2.8 Nm.

The housing surrounding the rotor was instrumented so that axial and radial displacement and resultant growth of the shaft and/or hub region could be detected during operation. In addition, bearing temperatures for both the rotor and the dynamometer were continuously monitored. A 3-axis accelerometer mounted on the turbine housing was used to detect any unusual vibrations.

Fig. 7.3. Dynamometer unit which was located below the expander (beneath floor level) during testing.

Shut down would have occurred automatically if low oil level or low oil vapor pressure occurred in the bearing lubrication system or if limits on torque, angular velocity, or displacement were exceeded. An audible alarm was used to warn the operator of excessive bearing temperature or acceleration.

Nozzle thrust, stagnation pressure, and pressure distribution at the exit of the nozzle were measured. Temperature and pressure were also measured at other selected locations within the rotor housing and exhaust ducting.

All data were recorded sequentially at a maximum rate of 0.3 seconds per channel on a 100 channel data logger. Permanent records were made on both paper tape (printed directly) and on magnetic tape (digital).