CFD ANALYSIS AND REDESIGN
OF CENTRIFUGAL IMPELLER FLOWS FOR ROCKET PUMPS

by

Alessandro Lupi

Thesis submitted to the Faculty of the
Virginia Polytechnic Institute and State University
in partial fulfillment of the requirements for the degree of
Master of Science
in
Mechanical Engineering

APPROVED:

[Signatures]
Dr. John Moore, Chairman

Dr. Clinton L. Dancey

Dr. Eugene F. Brown

December 1993
Blacksburg, VA
CFD ANALYSIS AND REDESIGN
OF CENTRIFUGAL IMPELLER FLOWS FOR ROCKET PUMPS

by

Alessandro Lupi

Dr. John Moore, Chairman

Mechanical Engineering

(ABSTRACT)

The analysis and redesign of a centrifugal impeller for a rocket pump is presented in this thesis. A baseline impeller was designed by Rocketdyne for the NASA Marshall Pump Consortium. Initially, the objective was to reduce the circumferential exit flow distortion of the baseline impeller. Later in the study, the objective became raising the head coefficient of the impeller. The study presented in this thesis was also undertaken to demonstrate current CFD capabilities for impeller design.

A literature review includes an overview of centrifugal impeller geometries and configurations. Centrifugal impeller performance and secondary flows are discussed, and a summary of studies on the effects of impeller exit and diffuser inlet velocity distortion on diffuser performance is also presented.

The flow calculation details and the results of the baseline impeller flow calculations are described. Fourteen redesigned impeller geometries were
analyzed using the Moore Elliptic Flow Program, and the results were compared to the baseline geometry in terms of head rise, losses, and exit flow distortions. A final geometry was chosen; this geometry will be built and tested by Rocketdyne.

The results show that backward blade lean can be effective in reducing the exit flow distortion of the impeller. Tip slots or holes were not beneficial because of the large inlet boundary layer. Also, it appears possible to raise the head coefficient of the baseline impeller without creating excessive flow distortion. The planned testing is necessary to verify the predictions of the flow code.
ACKNOWLEDGEMENTS

I would like to thank all those who helped me in developing and preparing this thesis. Also, I would like to thank all those people who contributed in making my 18 month stay in Blacksburg an extremely pleasant experience.

First of all, I would like to express my most sincere appreciation to Dr. John Moore, my graduate advisor, and to Joan G. Moore, his wife, for giving me the opportunity to work on this very exciting and challenging project, and for their guidance and enthusiasm. Also, I would like to thank Dr. Clinton L. Dancey and Dr. Hal L. Moses, who were also on my advisory committee. In addition, I would like to thank Dr. Eugene F. Brown for sitting in for Dr. Moses.

The list would not be complete without mentioning my wife, Kathy, who was always extremely supportive, and all my family and friends.
# TABLE OF CONTENTS

1.0 INTRODUCTION ................................................. 1

2.0 LITERATURE REVIEW - ROCKET PUMP DESIGN .......... 7
   2.1 Centrifugal Pump Configuration .................... 7
   2.2 Inlet Flow Distribution ................................ 16
   2.3 Design Parameters and Requirements ............... 18
   2.4 Pressures and Secondary Flows ................. 22
      2.4.1 Pressures in a Rotor .................. 22
      2.4.2 Secondary Flows in a Rotor ............. 23
   2.5 Diffuser Performance .................................. 27
   2.6 Methods for Changing Impeller Exit Flow Distribution 32
   2.7 The Moore Elliptic Flow Program .......... 34

3.0 OBJECTIVES OF THE REDESIGN .................. 36

4.0 RESULTS OF 3-D FLOW CALCULATIONS ............ 41
   4.1 3-D Flow Calculation Detail .................. 41
   4.2 Baseline Geometry .................................... 47
      4.2.1 Overall Performance .................. 47
      4.2.2 3-D Flow Development .............. 50
### LIST OF ILLUSTRATIONS

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Figure 1.1</td>
<td>Space Shuttle Main Engine</td>
<td>2</td>
</tr>
<tr>
<td>Figure 1.2</td>
<td>SSME Rocket Pump</td>
<td>3</td>
</tr>
<tr>
<td>Figure 1.3</td>
<td>Consortium baseline impeller</td>
<td>5</td>
</tr>
<tr>
<td>Figure 2.1</td>
<td>Experimental inducer cavitation characteristics</td>
<td>9</td>
</tr>
<tr>
<td>Figure 2.2</td>
<td>Basic inducer types</td>
<td>10</td>
</tr>
<tr>
<td>Figure 2.3</td>
<td>Rocket pump inducer</td>
<td>11</td>
</tr>
<tr>
<td>Figure 2.4</td>
<td>Impeller with inlet that controls cavitation</td>
<td>12</td>
</tr>
<tr>
<td>Figure 2.5</td>
<td>Consortium impeller test configuration</td>
<td>13</td>
</tr>
<tr>
<td>Figure 2.6</td>
<td>SEP turbopump</td>
<td>14</td>
</tr>
<tr>
<td>Figure 2.7</td>
<td>Absolute tangential and axial velocity distributions upstream of Consortium impeller</td>
<td>17</td>
</tr>
<tr>
<td>Figure 2.8</td>
<td>Impeller shape related to specific speed</td>
<td>19</td>
</tr>
<tr>
<td>Figure 2.9</td>
<td>Contours of reduced static pressure, $P_r$, and rotary stagnation pressure, $P^*$, in the radial part of a centrifugal impeller passage</td>
<td>24</td>
</tr>
<tr>
<td>Figure 2.10</td>
<td>Effects of blade lean on reduced static pressure distributions, $P_r$, and secondary flows</td>
<td>26</td>
</tr>
<tr>
<td>Figure 2.11</td>
<td>Impeller exit velocity distortion</td>
<td>29</td>
</tr>
<tr>
<td>Figure 2.12</td>
<td>$C_p$ vs. $B_t$ for vaneless diffuser</td>
<td>30</td>
</tr>
<tr>
<td>Figure 2.13</td>
<td>$C_p$ vs. $B_t$ for vaned diffuser</td>
<td>31</td>
</tr>
<tr>
<td>Figure 3.1</td>
<td>Pump Stage Team activities schedule</td>
<td>38</td>
</tr>
</tbody>
</table>
Figure 4.18 Calculation la: near-hub and hear-shroud velocity vectors 72
Figure 4.19 Calculation la: diffuser inlet plane flow distortion 73
Figure 4.20 Calculation lb: P, and P* on cross-sectional planes 75
Figure 4.21 Calculation lb: diffuser inlet plane flow distortion 76
Figure 4.22 Blade lean: geometries la, Rk-a, and Rk-b 78
Figure 4.23 Calculation Rk-a: P, and P* on cross-sectional planes 79
Figure 4.24 Calculation Rk-b: P, and P* on cross-sectional planes 80
Figure 4.25 Velocity distortion parameters vs. blade lean at impeller exit 83
Figure 4.26 Velocity distortion parameters vs. blade lean at diffuser inlet 84
Figure 4.27 Determination of geometry lc 85
Figure 4.28 Calculation lc: P, and P* on cross-sectional planes 86
Figure 4.29 Calculation lc: diffuser inlet plane flow distortion 87
Figure 4.30 Calculation ld: P, and P* on cross-sectional planes 89
Figure 4.31 Meridional view comparison: baseline and high head geometries 95
Figure 4.32 HH geometries: splitter blade leading edge meridional location 97
Figure 4.33 HH geometries: blade profile comparison at the shroud 98
Figure 4.34 Calculation HH 1: mass-averaged pressures 99
LIST OF TABLES

Table 4.1  Velocity distortion parameters at diffuser inlet  92
Table 4.2  Velocity distortion parameters at impeller exit  93
Table 4.3  High head geometries: overall performance parameters  115
Table 4.4  High head geometries: velocity distortion parameters  116
NOMENCLATURE

A = meridional distance parameter, Fig. 4.1

B_t = velocity distortion coefficient, Eq. 2.5.2

C = hub-to-shroud distance parameter, Fig. 4.1; area-averaged absolute velocity

C_p = pressure recovery coefficient

"du/dy" = effective velocity gradient, Eq. 2.7.4

g = acceleration due to gravity

H = head rise, Eq. 2.1.3

L = Prandtl mixing length, Eqs. 2.7.4 and 2.7.5

N_s = specific speed, Eq. 2.3.2

N_{ss} = suction specific speed, Eq.2.1.2

NPSH = net positive suction head, Eq. 2.1.1

p = static pressure

p_r = reduced static pressure, Eq. 2.4.2

p_t = total pressure, Eq. 2.4.1

p^* = rotary stagnation pressure, Eq. 2.4.3

p_v = vapor pressure

P, P^*, P_r, P_t = non-dimensional pressures, Eqs. 2.4.4 - 2.4.6

Q = volume flow rate
\( r, \theta, z = \text{cylindrical coordinates} \)

\( U_2 = \text{impeller tip speed} \)

\( u = \text{velocity} \)

"y" = distance to nearest wall, Eq. 2.7.5

\( \alpha = \text{absolute flow angle} \)

\((\alpha_{\text{max}} - \alpha_{\text{min}})_c = \text{circumferential flow distortion parameter, Eq. 2.5.2}\)

\((\alpha_{\text{max}} - \alpha_{\text{min}})_{\text{ax}} = \text{axial flow distortion parameter, Eq. 2.5.1}\)

\( \eta = \text{impeller efficiency} \)

\( \mu = \text{viscosity} \)

\( \rho = \text{density} \)

\( \tau = \text{wall shear stress, Eq. 2.7.5} \)

\( \psi = \text{head coefficient, Eq. 2.1.3 and 2.3.1} \)

\( \omega = \text{angular velocity} \)

\textbf{subscripts}

1 = inlet

abs = absolute

l = laminar

max = maximum

min = minimum
r = radial; reduced static
rel = relative
t = total; turbulent
z = axial
θ = tangential
_ = vector
1.0 INTRODUCTION

At present, many large rocket boosters are of liquid propellant design. Liquid propellant rockets have many features that make them suitable for use as boosters. In fact, liquid fueled rocket engines can be throttled much more easily than solid-fuel rockets, and liquid fuels and oxidizers have been found to be the most highly energetic propellants. Figure 1.1 shows the Space Shuttle Main Engine, which is fueled by liquid hydrogen and liquid oxygen.

One of the main problems encountered in devising liquid-propellant boosters is the design of a rocket pump that would supply the necessary fuel to oxidizer ratio to give maximum exhaust velocity, and to do so with such accuracy that the fuel and oxidizer tanks approach depletion at the same time (1). Rocket pumps are quite complicated systems. The Space Shuttle Main Engine (SSME) rocket pump, for example, consists of two low pressure and two high pressure pumps (Fig. 1.2). The liquid hydrogen and the liquid oxygen pumps have a low pressure turbopump installed upstream of a high pressure turbopump.

Many different parameters must be taken into consideration when designing a centrifugal pump. The pump must operate in a particular flow rate range and provide a certain stagnation pressure rise with a fluid of a given density. In addition, the efficiency must be high and the size should be as
SSME SETS STANDARD OF HIGH PERFORMANCE

- **HIGH ALTITUDE THRUST**: 512,000 LBS
- **CHAMBER PRESSURE**: 3200 PSIA
- **SPECIFIC IMPULSE AT ALTITUDE**: 455 SECONDS
- **THROTTLE RANGE**: 65 TO 109%
- **PROPELLANTS**: OXYGEN/HYDROGEN
- **WEIGHT**: 7000 LBS
- **DESIGN LIFE**: 27,000 SECONDS
  55 STARTS

**Figure 1.1** Space Shuttle Main Engine (3)
FOUR TURBOPUMPS REQUIRED TO ACHIEVE HIGH PRESSURES

Figure 1.2. SSME Rocket Pump (3)
small as possible but consistent with good efficiency and the required pressure rise (2).

The centrifugal impeller is a very important element in a rocket pump, since the merit of the impeller design reflects on the overall performance of the pump stage. Among the factors that affect the performance of a centrifugal impeller, are the size, the speed of rotation, the number and shape of full blades and, if applicable, of splitter blades.

Computational methods for turbomachinery design and analysis have become more accurate and widespread over the past few years. In order to advance rocket propulsion technology, the Consortium for Computational Fluid Dynamics (CFD) has been formed at the NASA Marshall Space Flight Center. The consortium consists of three separate teams: the Turbine Stage Team, the Pump Stage Team, and the Combustion Devices Team. The Pump Stage Team has implemented a plan for pump technology development which will produce validated CFD codes suitable for application to pump components, test data useful for validating CFD codes, and state of the art technology pump components optimized using CFD codes (4). A Consortium baseline impeller geometry, shown in Fig. 1.3, was designed by Rocketdyne using a combination of traditional and CFD techniques.

The next section of this thesis gives a brief literature review of previous studies and tests conducted on rocket pumps and centrifugal compressors. It
also introduces important terms that will be used throughout the thesis.

The objectives of the redesign of the Consortium impeller will be discussed in detail in the following section. This section also states the specific objectives of the VPI & SU team and of the author of this thesis.

The results of the flow calculations will be presented in section 4 of the thesis. The section will present the many geometries that were analyzed and the design process that was followed to determine the final geometry.

Finally, section 5 will discuss important findings and conclusions that were made as a result of this study. The merits of CFD methods for turbomachinery design and analysis will also be discussed in this section.
2.0 LITERATURE REVIEW - ROCKET PUMP DESIGN

2.1 Centrifugal Pump Configuration

Several different centrifugal pump configurations are possible in turbomachinery design. The simplest configuration is the single stage centrifugal pump with no upstream inducer. When a high pressure rise needs to be achieved, however, multistage pumps are often used. In multistage pumps the flow exiting each stage is fed into the following stage through a turnaround duct.

Inducers are used when the inlet flow is likely to cavitate. Cavitation is the spontaneous formation of vapor bubbles in a liquid as the static pressure falls below the vapor pressure. One of the most important requirements in the design of rockets is low vehicle weight. In order to minimize the weight the fuel tank walls are built relatively thin, which does not allow them to withstand high pressures. The low tank pressure translates into a low value of the net positive suction head (NPSH), defined as:

\[
\text{NPSH} = (p_{11} - p_v) / \rho g \tag{2.1.1}
\]

The suction specific speed of the pump is given by:

\[
N_{ss} = \omega Q^{1/2} / (g \text{ NPSH})^{3/4} \tag{2.1.2}
\]

The pressure rise or head coefficient, \( \Psi \), is defined as:

\[
\Psi = gH/U_2^2 \tag{2.1.3}
\]

Since rocket pumps are required to have a high head rise, \( H \), and the head
coefficient is limited by flow slip, friction losses, and flow separations, the tip speed \( U_2 \) must also be large. Therefore, the angular velocity must be large. In combination with a low NPSH, this creates a high \( N_{ss} \). The effect of high \( N_{ss} \) on the cavitation characteristics of an inducer is shown in Fig. 2.1. At high \( N_{ss} \) (low NPSH), the cavitation is widespread, resulting in a lower head rise. At low \( N_{ss} \), the cavitation is limited to a tip vortex.

Inducers are axial flow pumps whose task is to pressurize fluid whose static pressure is not much higher than its vapor pressure (2). Figure 2.2 shows some of the basic inducer types. Typical rocket pump inducers have thin, sharp-edged, flat blades formed in spirals around the hub (Fig. 2.3). Some impellers are designed with inlets that control cavitation, where the inducer blading is continuous with the main impeller blading (Fig. 2.4). These designs, however, do not exhibit very good performance at high suction specific speeds (6).

Many rocket pumps have a separate inducer and impeller that are installed on the same shaft. This arrangement is used for both single stage and multistage pumps. An example of this arrangement is the Consortium impeller in the test configuration (Fig. 2.5). The Société Européenne de Propulsion (SEP) rocket pump shown in Fig. 2.6 features a two stage pump with an upstream inducer. The inducer and the impeller are mounted on the same shaft, but are separated by stator vanes.
Figure 2.1 Experimental inducer cavitation characteristics (3)
Figure 2.2 Basic inducer types (5)
Figure 2.3  Rocket pump inducer (2)
Figure 2.4 Impeller with inducer that controls cavitation (6)
Figure 2.6 SEP Turbopump
Finally, some rocket pumps have separate inducers and impellers which are mounted on separate shafts. This type of set-up is used in the SSME fuel pump, where there is an axial flow pump upstream of a three stage centrifugal pump but mounted on a different shaft (Fig 1.2). The SSME High Pressure Fuel Turbopump used three stages to maximize specific speed within rotating speed and critical speed constraints imposed by the rotating assembly length. The three identical impellers are shrouded so that the pump performance is not sensitive to the housing deflections resulting from the very high pressure level (7).
2.2 Inlet Flow Distribution

The inlet flow distribution in centrifugal pumps varies depending on the particular configuration. If the impeller has no upstream inducer and a short inlet duct, the inlet flow will be nearly uniform; with a long inlet duct the inlet velocity profile will approach the fully developed pipe flow distribution. On the other hand, the presence of an upstream inducer or a turnaround duct will significantly distort the inlet flow. The Consortium impeller, for example, has a strongly nonuniform inlet flow since the inducer tends to move the high loss fluid towards the shroud region. Figure 2.7 shows tangential and axial velocity contours upstream of the Consortium impeller. The velocities were measured by Rocketdyne using laser velocimetry, and were made non-dimensional with the impeller tip speed.
Figure 2.7  Absolute tangential and axial velocity distributions upstream of Consortium impeller
2.3 Design Parameters and Requirements

Centrifugal pumps should efficiently add work to an inlet flow which may be greatly distorted, and create an acceptable exit flow. Overall performance parameters should meet the design requirements, and the exit flow should be within some specifications in terms of uniformity.

There are some important parameters that are used in the design and analysis of centrifugal pumps. The head coefficient is a non-dimensional parameter that measures the pressure rise across the pump, and is given by:

\[ \Psi = \frac{gH}{\omega^2 r_2^2} = \frac{\Delta p_1}{\rho U_2^2} \]  \hspace{1cm} [2.3.1]

Typical head coefficients for rocket pumps vary from 0.5 to 0.6. The head coefficient of the redesigned Consortium impeller varied from 0.6 to 0.7.

Another important non-dimensional parameter is the specific speed. The non-dimensional specific is defined as:

\[ N_s = \frac{\omega Q^{1/2}}{(gH)^{3/4}} \]  \hspace{1cm} [2.3.2]

When English units are used, the specific speed is usually given by:

\[ N_s = \frac{N(rpm)Q(gpm)^{1/2}}{(H(ft))^{3/4}} \]  \hspace{1cm} [2.3.3]

The overall shape of the impeller can vary greatly depending on the value of its specific speed. The exit passage height, for example, for optimum impellers, becomes larger as the specific speed of the impeller increases. Figure 2.8 shows how the shape and the performance characteristics of typical impellers vary with the specific speed. The specific speed can be converted from the
Figure 2.8 Impeller shape related to specific speed
British units used in the figure to non-dimensional values by dividing by 2730. It must be observed that the figure refers to commercial centrifugal pumps, which tend to have a radial flow inlet. Rocket pumps usually have an axial flow inlet. The non-dimensional specific speed of typical rocket pump impeller ranges from 0.22 to 0.55. The specific speed of the baseline Consortium impeller is 0.4.

Size is another important parameter in centrifugal impeller design. The tip speed of the impeller, for example, is a function of the rotational speed as well as the impeller diameter. The impeller passages must also be large enough to accommodate the desired flow rate. The size of the impeller is characterized by its axial length, the exit diameter, the inner and outer inlet diameter, and the exit passage height. The shape of the impeller blades is affected by the blade lean, the discharge blade angle distribution, and the blade wrap. The blade lean is the angle that the blade forms with the normal to the hub surface on quasi-orthogonal planes. The blade is leaned backward if it is tilted towards the suction side, forward if it is tilted towards the pressure side. Blade lean and its effects on the impeller flow are explained in detail in section 2.4.2. The discharge blade angle is the angle formed by the impeller blade and the tangent to the impeller disk at the exit. This angle equals 90 degrees when the impeller blade is radial at the exit. Rocket pumps often have impellers with backward swept blades, as shown in Fig. 1.3, with discharge blade angles less than 90
degrees. The discharge blade angle can be constant or vary as a function of the passage height, depending on the design of the impeller. The blade wrap is the difference in tangential position of the blade between the inlet and the exit of the impeller for a given passage height.

Several rocket pumps have splitter blades. Splitter blades are used when it is desirable to use the axial portion of the pump to guide the flow, and to do most of the work in the radial portion of the pump. The Consortium impeller, for example, has one set of splitter blades, while the SSME HPFTP has two sets of splitter blades. Flow split is an important parameter in rocket pumps with splitter blades. The flow split measures the amount of mass flow rate that is channeled between the full blade suction side and the splitter blade pressure side compared with the mass flow that is channeled between the splitter blade suction side and the full blade pressure side for one full passage.
2.4 Pressures and Secondary Flows

2.4.1 Pressures in a Rotor

For incompressible flows with density $\rho$, the total pressure is given by the equation:

$$p_t = p + .5\rho u_{abs}^2$$  \hspace{1cm} [2.4.1]

When the flow is analyzed in a rotating reference frame, such as in turbomachinery applications, it is useful to define the reduced static pressure, $p_r$, and the rotary stagnation pressure, $p^*$:

$$p_r = p - .5\rho w^2 r^2$$  \hspace{1cm} [2.4.2]

$$p^* = p + .5\rho w^2 r^2 - .5\rho w^2 r_2^2$$  \hspace{1cm} [2.4.3]

The low $p^*$ fluid is often referred to as high loss fluid. In the MEFP calculations, $p_r$ and $p^*$ have been normalized with $
.5\rho w^2 r_2^2$

The reference pressure is taken as the maximum value of the rotary stagnation pressure on the inlet plane for the calculation,

$$p^*_{\text{max, inlet}}$$

Therefore, the non-dimensional pressures are:

$$P_t = (p_t - p^*_{\text{max, inlet}})/.5\rho w^2 r_2^2$$  \hspace{1cm} [2.4.4]

$$P_r = (p_r - p^*_{\text{max, inlet}})/.5\rho w^2 r_2^2$$  \hspace{1cm} [2.4.5]

$$P^* = (p^* - p^*_{\text{max, inlet}})/.5\rho w^2 r_2^2$$  \hspace{1cm} [2.4.6]

The mass-averaged values of the total, static, and rotary stagnation
pressure are also helpful in analyzing a centrifugal pump. The mass-averaged total pressure gives a measure of the head rise of the flow. The mass-averaged static pressure shows how the static pressure rise is achieved as the flow proceeds in the meridional direction. The rotary stagnation pressure, on the other hand, gives the development of the losses as a function of the meridional distance.

2.4.2 Secondary Flows in a Rotor

Distributions of reduced static pressure and rotary stagnation pressure can also be used to help understand the development of secondary flows in a rotor. Consider for example a cross-section in the radial part of a centrifugal impeller as shown in Fig. 2.9. Contours of $P_r$ and $P^*$ are shown, and a region of high loss fluid is located near the shroud wall. The secondary flow of this high loss fluid from the pressure side to the suction side can be considered to be due to Coriolis effects. The circumferential gradient of reduced static pressure is established by the Coriolis acceleration, $2w_r\Omega$, of the bulk of the flow. This pressure gradient is then imposed on the lower velocity, high loss fluid (with a lower Coriolis acceleration) near the shroud. The net force on the fluid results in a convection of this fluid towards the suction side.

In general, to a first approximation, high loss fluid is driven by the gradients of reduced static pressure and tends to seek regions of low $P_r$. It also
Figure 2.9 Contours of reduced static pressure, $P_r$, and rotary stagnation pressure, $P^*$, in the radial part of a centrifugal impeller passage
can continue its motion past the location of the minimum $P_r$, carried by its own inertia.

Blade lean can be used to reorient the impeller blades relative to the reduced pressure field. This is because the static pressure field remains nearly fixed in space while the change in location of the blade occurs. Thus it is possible using blade lean to move the location of the minimum reduced static pressure. Figure 2.10 shows cross-sections of two impeller passages, one with blade lean and one without. In this thesis, the blade lean angle is defined as the angle the blade makes to the normal to the hub in an orthonormal plane. Thus the impeller shown in Fig. 2.10b has the backward lean angle shown. Note that the contours of reduced static pressure are relatively unchanged between Figs. 2.10a and 2.10b. So the effect of blade lean has been to move the minimum pressure from the shroud/suction-side corner in Fig. 2.10a to the hub/suction-side corner in Fig. 2.10b. In the impeller of Fig. 2.10a, high loss fluid near the shroud wall would tend to remain near the shroud, while in the impeller of Fig. 2.10b, that fluid would tend to be convected towards the hub along the suction side.
Figure 2.10 Effects of blade lean on reduced static pressure distributions, $P_r$, and secondary flows. Curved arrows show secondary flow paths of high loss boundary layer fluid.
2.5 Diffuser Performance

The impeller exit flow has an effect on the diffuser performance, and therefore on the performance of the pump as a whole. In general, the efficiency of centrifugal impellers is about 95 percent. The stage efficiency, however, is typically on the order of 80 percent. Therefore, the diffuser is the limiting factor to achieving high stage efficiencies. A lot of attention must be given to the diffuser inlet flow distribution.

Some parameters have been defined and give a measure of the diffuser inlet flow distortion. Rocketdyne uses the parameter $C^2(\sigma_{\text{max}} - \sigma_{\text{min}})$, where $C$ is the area-averaged absolute velocity and $\sigma$ is the axially area-averaged absolute flow angle. Moore, Moore, and Lupi (10) observed that the term $C^2$ is roughly the same for impellers of the same family and can therefore be omitted. They used the parameters

$$a_{\text{max}} - a_{\text{min}}$$  \hspace{1cm} [2.5.1]

$$a_{\text{max}} - a_{\text{min}}$$  \hspace{1cm} [2.5.2]

The two parameters represent the difference between the maximum and the minimum values of the circumferentially area-averaged and hub-to-shroud area-averaged absolute flow angle, respectively. These parameters therefore represent the flow distortion in the axial and in the circumferential directions, respectively.

Yoshinaga et al. (11) observed that the non-uniformity of the flow angle

27
at the diffuser inlet plane is due mainly to the radial velocity distortion (Fig. 2.11). They therefore defined the velocity distortion coefficient $B_r$, given by the ratio of the mass-averaged radial velocity to the area-averaged radial velocity:

$$B_r = \frac{u_{r, \text{mass-avg}}}{u_{r, \text{area-avg}}} \quad [2.5.3]$$

Yoshinaga conducted experiments on both vaneless diffusers and diffusers with half guide vanes, and his results showed that the pressure recovery across the diffuser varies inversely with the velocity distortion coefficient (Fig. 2.12). A few years later Agrawal et al. (12) demonstrated that the same trend exists in vane island diffusers. They also observed that as the velocity distortion coefficient increases, the pressure recovery coefficient decays with $B_r$ (Fig. 2.13). Both Yoshinaga and Agrawal found that the distortion in the circumferential direction decays quickly between the impeller exit and the diffuser inlet planes. The distortion in the axial direction, on the other hand, tends to remain constant, and is therefore the main contributor to the overall diffuser inlet flow distortion.
Figure 2.11 Impeller exit velocity distortion (11)
Figure 2.12 $C_p$ vs. $B_f$ for vaneless diffuser (11)
Figure 2.13 $C_p$ vs. $B_t$ for vaned diffuser (12)
2.6 Methods for Changing Impeller Exit Flow Distribution

The impeller exit flow distribution can be improved by modifying the impeller geometry. Goto (13) obtained improvements to the exit flow distortion of a mixed-flow pump by cutting small slots along the second portion of the blade near the shroud wall. The flow through the slots blew the high loss regions from the suction side to the middle of the passage, thus preventing accumulations of high loss fluid at the suction side of the blade near the shroud wall. The resulting flow was more uniform. The inlet boundary layer in Goto’s experiments measured 25-30 percent of the passage height.

Moore, Moore, and Lupi (10) obtained improvements in the exit flow distribution of the baseline Consortium impeller in their calculations by leaning the impeller blades. The baseline impeller blades were leaned backward (toward the suction side) starting after the splitter blade leading edge. The change in geometry was found to create a minimum reduced static pressure region in the hub/suction-side corner, towards which high loss fluid was convected by secondary flows. The end result was a more uniform loss distribution in the circumferential direction, and, consequently, lower circumferential velocity distortion at the exit.

The location of the splitter blade leading edge can also influence the exit flow distribution. The tangential location of the splitter blade leading edge can affect the flow split. An uneven flow split causes different pressure
distributions in the two half passages and increases the distortion of the exit flow.
2.7 The Moore Elliptic Flow Program

The flow calculations were made using the Moore Elliptic Flow Program, or MEFP. MEFP is a 3-D pressure correction solution procedure for the Navier-Stokes equations. The program was developed as a general code for three-dimensional steady flows in turbomachinery blade rows. The code has been used in several different applications. For instance, it has been utilized to analyze flows in axial-flow turbines, axial inducers, as well as centrifugal compressors. The discretization of the conservation equations and the solution procedure have been described by J.G. Moore (14).

The continuity and momentum equations, in a rotating reference frame for steady incompressible flow, can be written in the form:
\[ \nabla \cdot \rho u = 0 \]  \hspace{1cm} (2.7.1)
and
\[ \rho u \cdot \nabla u - \nabla \cdot \mu \nabla u = \nabla p - 2\rho w x u - \rho w x (\omega x) \] \hspace{1cm} (2.7.2)

The effective viscosity, \( \mu \), is modeled using a Prandtl mixing length viscosity model with local searches for boundary and shear layer widths. In the near-wall regions, a van Driest correction is applied to the mixing length. A near-wall correction is also applied to the calculated effective viscosity, in order to reduce the sensitivity of the turbulent boundary layer calculations to the near-wall grid spacing. The turbulence model is described by the equations (15):
\[ \mu = \mu_f + \mu_t \] \hspace{1cm} (2.7.3)
\[ \mu_t = \rho L^2 \frac{\text{du/dy}}{} \]  

[2.7.4]

where \( L \) is the smaller of:

- 0.08 times the width of shear or boundary layer
- 0.41 times the distance to the nearest wall

The van Driest correction is given by the following equation:

\[ L = 0.41 \frac{\text{y}}{\text{y} \exp\left(-\frac{\text{y} (\rho \tau)^{1/2}}{26 \mu_t}\right)} \]  

[2.7.5]

The near-wall correction is:

\[ \mu = [\mu_t (\mu_t + \mu_t)]^{1/2} \]  

[2.7.6]
3.0 OBJECTIVES OF THE REDESIGN

In late 1989 it became clear at the NASA Marshall Space Flight Center (MSFC) that very significant advances had been made in CFD analysis techniques. Whereas the design of some rocket engine components had benefitted from advances in CFD techniques, the design approach and, consequently, the performance of rocket pumps had not changed significantly over the past 20 years. In order to encourage the industry to implement CFD methodologies in their design process, MSFC decided to organize a program that would include code validation, application of these codes to state-of-the-art hardware concepts, and verification of the concepts through testing. The codes and the hardware were selected so as to benefit the development of NASA’s next generation rocket engine, the Space Transportation Main Engine (STME).

As a result, the consortium for CFD Application in Propulsion Technology was formed at MSFC in 1990 to accomplish the fore-mentioned objectives. The consortium consists of three teams: the Turbine Stage Team, the Pump Stage Team (PST), and the Combustion Device Team. Each team has a NASA and an industry co-chairman. Dr. P. McConnaughey is the overall consortium manager and Mr. R. Garcia and Mr. A.H.J. Eastland are co-chairmen for the PST.

The Pump Stage Team has three specific objectives. The first objective
is to coordinate MSFC sponsored activities aimed at the advancement, application, and demonstration of CFD technology for pump design. Secondly, the PST should provide a forum for interaction and technology transfer among pump design and analysis experts. Finally, the PST should provide a peer review for all activities in the MSFC PST Program.

Figure 3.1 shows a schedule that has been implemented to coordinate the team’s activities and set milestone dates consistent with the needs of the STME Program. The program defined in the figure is focused on the developments suitable for the use in the SSME fuel pump. The current SSME fuel pump design needs three stages to meet the engine system requirements. A two stage design was considered too large of a risk in the SSME program. The initial activities of the PST have been centered on using CFD to design an impeller that would allow for a two stage pump design (4).

The new impeller design should have fewer splitter blades and would therefore be lighter, and should also be more durable and easier to manufacture. The impeller redesign project was divided into two phases. Each of the two phases was focused on obtaining certain flow characteristics. During the first phase, the focus was on improving the impeller exit flow distribution. The purpose of this phase was to minimize the diffuser vane excitation, which affects durability, and to improve the pressure recovery across the diffuser. In the second phase of the design process the emphasis
### PUMP DESIGN TECHNOLOGY

<table>
<thead>
<tr>
<th>Activities</th>
<th>Fiscal Year 1991</th>
<th>Fiscal Year 1992</th>
<th>Fiscal Year 1993</th>
<th>Fiscal Year 1994</th>
<th>Fiscal Year 1995</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>4th 1st 2nd 3rd</td>
<td>4th 1st 2nd 3rd</td>
<td>4th 1st 2nd 3rd</td>
<td>4th 1st 2nd 3rd</td>
<td>4th 1st 2nd 3rd</td>
</tr>
<tr>
<td>StiME Program</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>LLDR/PDR</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>CDR</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TurboPump Test</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Advanced Hardware</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Baseline Impeller</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Conventional Design</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>CFD Analysis</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Manufacture</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Water Test</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Phase 1 Impeller</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>CFD Parameters</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Design</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>CFD Analysis</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Manufacture</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Water Test</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Validation Data</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Inducer (ϕ = 0.3)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SSME Impeller</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Baseline Impeller</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Inducer (ϕ = 0.08)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Diffuser Data 1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Code Development</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>AEROVISIC</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>FDNS3D</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>HAH3D</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>INSD3D-UP</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>MINT</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>REACT3D</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>MERR</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Figure 3.1** Pump Stage Team activities schedule (4)
changed from improving the exit flow distribution to raising the head coefficient from 0.6 to 0.7. A very important goal of the project undertaken by the PST was to study the feasibility and the benefits of using Computational Fluid Dynamics codes for turbomachinery design purposes compared to more traditional design methods. The final design, in fact, will be built and tested, and the experimental results will be compared with the CFD calculations results.

The objectives of the Virginia Polytechnic Institute and State University team were to support the Consortium PST effort by performing MEFP calculations and to suggest possible design improvements. The experimental data that will be collected by Rocketdyne during the testing of the final impeller design will give a measure of the accuracy of the MEFP flow code in high loss, incompressible flow applications. The results will also provide insight on the precision of the physical modeling and the merits of the upwind control volume approach in MEFP which allows highly non-uniform grid spacing and relatively coarse grids to be used in the calculations.

The objectives of the work described in this thesis were to analyze and evaluate the results of a series of 3-D viscous code calculations for a range of centrifugal impeller designs, and to perform a literature study on centrifugal pump vaned diffuser performance. In addition, the goal was to quantify the performance of different impeller designs in terms of, for example, flow
distortion, flow splits, and pressure distributions.
4.0 RESULTS OF 3-D FLOW CALCULATIONS

4.1 3-D Flow Calculation Detail

The flow field for the different impellers that were analyzed was calculated for a single passage, which consists of one full blade and one splitter blade. Figure 4.1 shows the calculation grid for the Consortium baseline impeller. The number of grid points varied in the hub-to-shroud direction from 13 to 19, and in the blade-to-blade direction from 23 to 25, depending on which geometry was analyzed. There were 69 to 75 grid points in the flow direction; of these, approximately ten grid points were upstream of the impeller inlet and approximately twenty downstream of the impeller exit.

Strongly nonuniform grid spacing was used. Grid spacing was smallest near the walls, where developing boundary layer effects need to be analyzed. Grid spacing varied from 0.2 percent of the passage height near the shroud wall to 14 percent of the passage height in the middle of the passage. A non-dimensional parameter was used to measure the distance in the flow direction. The parameter A is the sum of the axial and radial position along the impeller passage. This parameter is non-dimensionalized so that it equals 0 at the full blade leading edge, 1 at the splitter blade leading edge, and 2 at the impeller exit plane. Similarly, the parameter C was used to measure the position from hub to shroud. The parameter equals 0 at the hub and 1 at the shroud.

Figure 4.2 shows the overall dimensions of the baseline Consortium
Figure 4.1 Calculation grid for baseline Consortium impeller
Figure 4.2 Overall dimensions of the baseline Consortium impeller
impeller. The impeller is fully shrouded and has six full blades and six splitter blades. The angular velocity of the impeller at the design flow rate is 662.4 rad/s. The fluid chosen for the calculations is water. Water is also the fluid used by Rocketdyne to collect experimental data. The Reynolds number based on impeller exit tip speed and diameter is $1.7 \times 10^7$.

The inlet flow conditions used for the calculations are shown in Fig. 4.3. The axial and tangential inlet velocity profiles were obtained by averaging the experimental data shown in Fig. 2.7 in the circumferential direction. The radial velocity was set to zero and the hub-to-shroud static pressure variation was determined from radial equilibrium. A mass flow rate of 12.33 kg/s per passage was used for the calculations. This flow rate was determined from the measured inlet velocity profile, and corresponds to an impeller flow rate of 74.0 kg/s.

For the wall boundary conditions, zero relative velocity was imposed on the rotating hub upstream of the impeller and on the impeller walls, including the rotating shroud. Zero absolute velocity was imposed on the stationary shroud wall upstream of the impeller leading edge.

The calculation was run with an inviscid wall vaneless diffuser. A vane island diffuser with an inlet at a radius of 5.0 inches (see Fig. 4.2) is planned for the experimental data. This is of interest in the present study, as it is at this diffuser inlet plane that the impeller exit flow distortion is to be minimized.
Figure 4.3 Inlet flow conditions; velocities normalized with the impeller tip speed; dimensionless rotary stagnation and reduced pressures.
All calculations were performed on a Hewlett Packard 720 workstation. In order to ensure convergence, one hundred iterations were executed for each calculation. The average run time was approximately 18 hours of computer time.
4.2 Baseline Geometry

The objective of the design of the Consortium baseline impeller was to meet the Space Transport Main Engine fuel pump requirements by using two stages instead of the three used in the present SSME design. The baseline impeller is a conventional radial impeller of low specific speed. It was designed with the conventional design methodology of Rocketdyne, and was subsequently optimized through a CFD parametric study. Since the efficiency and the head coefficient were adequate after the conventional design stage, the impeller was optimized mainly for minimum impeller exit flow distortion (4).

Figure 1.3 shows a three dimensional view of the baseline impeller. The meridional and blade-to-blade views can be observed in Fig 4.1, and the overall dimensions are given in Fig. 4.2.

4.2.1 Overall Performance

Figure 4.4 shows the development of the dimensionless mass-averaged pressures as a function of the meridional distance parameter A. The total and static pressures rise slowly until the splitter blade leading edge. After the flow becomes mostly radial, the static and total pressures rise sharply until the impeller exit. In the vaneless diffuser, the total pressure remains relatively constant (since no more work is done on the fluid and an inviscid wall boundary condition was used for the vaneless diffuser), whereas the static pressure
Mass-averaged pressures
  t-total, s-static, *-rotary stagnation, r-reduced

\[ \delta P_t = 1.417 - 0.195 = 1.222 \]
\[ \delta P^* = -0.053 - (-0.026) = -0.027 \]
Eff = 98 %

Figure 4.4 Baseline calculation: mass-averaged pressures
recovery continues.

Figure 4.4 gives the values of the dimensionless total pressure, \( P_t \) (Eq. 2.4.4) as 1.417 at the impeller exit and 0.195 at the impeller inlet. Thus the rise in total pressure through the impeller is \( \delta P_t = 1.222 \).

Figure 4.4 also shows the increase in losses (-\( \delta P^* \)) through the impeller. The losses are quite small (0.027) in the present calculation; in fact, they are very close to the losses produced in the upstream inducer (0.026).

From the results at the impeller exit in Fig. 4.4, the impeller head coefficient can be calculated.

\[
gH/U_2^2 = \Delta p_t/\rho U_2^2 = \delta P_t/2 = 0.611
\]

The efficiency is given by

\[
\eta = \delta P_t/(\delta P_t + (\delta P^*)) = 1.222/(1.222 + 0.027) = 97.8\%
\]

Also, the specific speed of the impeller can be calculated from Eq. 2.3.3,

\[
N_* = N(\text{rpm}) \left[ Q(\text{gpm}) \right]^{1/2} / \left[ H(\text{ft}) \right]^{3/4}
\]

using

\[
H = \delta P_t \cdot \rho U_2^2 / 2\rho g = 1.222 \left( 248.26 \text{ ft/s} \right)^2 / 2 \cdot 32.2 \text{ ft/s}^2
\]

or

\[
H = 1169.5 \text{ ft}
\]

and with \( N = 6325.4 \) rpm and \( Q = 1174.3 \) gpm,

\[
N_* = 6325.4 \text{ rpm} \left[ 1174.3 \text{ gpm} \right]^{1/2} / \left[ 1169.5 \text{ ft} \right]^{3/4}
\]

\[
N_* = 1083.9 \text{ rpm gpm}^{1/2} / \text{ft}^{3/4}
\]

or

\[
N_* = 0.397 \text{ dimensionless.}
\]
4.2.2 3-D Flow Development

Figure 4.5 shows development of the reduced pressure and rotary stagnation pressure at four quasi-orthogonal planes along one impeller passage. At the inlet, the reduced static pressure has a hub-to-shroud variation, with high pressure at the hub and low pressure on the shroud, see Fig. 4.3. As the full blade is loaded due to rotational effects, the low reduced static pressure moves to the shroud/suction-side corner of the passage, while the high pressure moves to the hub/pressure-side corner. Finally, as the flow turns to the radial direction towards the exit, the variation becomes blade-to-blade, before unloading at the exit.

The rotary stagnation pressure also has a hub-to-shroud variation at the impeller inlet. This is caused by the inducer exit flow, see Fig. 4.3. The high loss fluid is spread across the shroud region. The high loss fluid remains near the shroud until after the splitter blade leading edge. At that point, secondary flows begin to move the high loss fluid towards the shroud/suction-side corner, where the reduced static pressure has its lowest value. Near the impeller exit, the inertia of the secondary flows carries part of the high loss fluid towards the hub. The convection of this high loss fluid can be followed by observing the contours of $\delta P^* = -0.1$ at successive planes.

The near-blade and near-hub and near-shroud velocity vectors are shown in Fig. 4.6 and Fig. 4.7, respectively. The flow appears to be well behaved and
Figure 4.5 Baseline calculation: $P_r$ and $P^*$ on cross-sectional planes. $s$-suction side of full blade; $p$-pressure side of splitter blade.
Figure 4.6 Baseline calculation: near-blade velocity vectors
Figure 4.7 Baseline calculation: near-hub and near-shroud velocity vectors; velocity vectors for all heights at the impeller exit showing the exit flow angles.
backflow occurs only at the full and splitter blade leading edges near the shroud wall. Along the suction side of both the full blade and the splitter blade, the vectors show the convection of the high loss fluid from the shroud wall towards the hub, downstream of the splitter blade leading edge. Correspondingly, along the pressure side of the blades, secondary flows move towards the shroud wall, enhanced by the effects of meridional curvature. The near-shroud velocity vectors also show high loss fluid moving from the pressure side towards the suction side.

4.2.3 Impeller Exit Flow

The velocity vectors at the impeller exit, Fig. 4.7, show very stratified blade-to-blade flow. The measures of distortion of this flow are discussed in Section 2.5. These are the parameters \((\alpha_{\text{max}}-\alpha_{\text{min}})_c\), \((\alpha_{\text{max}}-\alpha_{\text{min}})_h\), and \(B_i\). The values of these parameters and their decay downstream of the impeller exit are shown in Fig. 4.8. The overall distortion and the circumferential distortion parameters decay as a function of the radius downstream of the impeller exit; however, the hub-to-shroud distortion parameter continues to increase throughout the diffuser.

The overall distortion parameter \(B_i\) falls from a value of 1.53 to at the impeller exit to 1.03 at the location of the proposed vaned diffuser inlet. As discussed in section 2.5, Agrawal et al. only found decreases in vaned diffuser
Figure 4.8 Distortion parameters downstream of impeller exit.
$r/r$-exit = 1.11 = diffuser inlet plane.
performance for $B_t$ values above 1.07. Thus the vaned diffuser inlet distortion shown in Fig. 4.8 should not cause a reduction in pressure recovery. The circumferential distortion, $(\alpha_{\text{max}}-\alpha_{\text{min}})_c$, falls from 16.6 degrees at the impeller exit to 2.0 degrees at the vaned diffuser inlet, where the mean flow angle is 6.2 degrees. Therefore there may be some reduction in diffuser vane excitation to be had by reducing this parameter.

The diffuser inlet plane distortion is shown in Fig. 4.9, which shows contours of rotary stagnation pressure, absolute velocity squared and tangent of the flow angle. The diffuser inlet flow is still distorted, with regions of high and low absolute velocity and very nonuniform absolute flow angle. The region of high absolute flow velocity, 0.6, in the middle of the flow corresponds to the wake or high loss region which came from the suction side of the full blade. The regions of low absolute flow velocity, near the shroud wall, are in the jets of low loss fluid which came from the shroud/pressure-side corner regions of the passages. The absolute flow angle varies from 2.9 degrees (contour 0.05) near the hub to 8.5 degrees (contour 0.15) between the jets and wakes in the middle of the flow.
Dimensionless $P^*$, $\delta P^* = 0.025$

Absolute velocity squared, $\delta C^2 = 0.025$

Tangent of absolute flow angle, $\delta C_r/C_\theta = 0.01$

Figure 4.9 Baseline calculation: diffuser inlet plane flow distortion; contours of $P^*$, dimensionless absolute velocity squared and tangent of the absolute flow angle.
4.3 First Phase of Impeller Redesign - Exit Flow Distortion Improvement

4.3.1 Reasons for Reducing Diffuser Inlet Flow Distortion

The first phase of the Consortium baseline impeller redesign was focused on improving the impeller exit flow blade-to-blade distortion, as seen in Fig. 4.7, and as measured by \((a_{\text{max}} - a_{\text{min}})_e\), shown in Fig. 4.8. The impeller exit flow distortion does not completely decay in the vaneless region that separates the impeller from the diffuser vanes. As a result, the incidence at the diffuser vane leading edge varies as a function of time and also from hub to shroud. The resulting transient excitation might affect the durability of the diffuser vanes. Since this excitation is mainly governed by the circumferential distortion at the diffuser inlet plane, the first phase redesign efforts were directed towards reducing this parameter.
4.3.2 Slots

The first attempts to redesign the baseline impeller were made by applying Goto’s findings (13) to the impeller design. Four different impeller geometries were created by introducing different size slots or holes on the shroud region of the blade after the splitter blade leading edge.

The geometries fa and fb both had tip slots measuring one percent of the blade height, beginning after the splitter blade leading edge. Geometry fa was a shrouded impeller, whereas geometry fb was unshrouded. In both cases, the tip slots were very narrow compared to the size of the high loss fluid region, see Fig. 4.3, and the flow patterns were similar to the baseline calculation. In the calculation fa, the presence of the tip gap caused an increase in the losses in the impeller by a factor of two, and reduced the work done on the fluid. The stationary shroud of geometry fb reduced the losses compared to calculation fa, but had no significant effect on the secondary flows.

Since small gaps were found to be ineffective, geometry fc was given a large tip slot, with a height equal to 10 percent of the blade height starting after the splitter blade leading edge (Fig. 4.10). The large slot caused secondary flows to move from the suction side to the pressure side across the shroud as desired (Fig. 4.11) and it kept some of the high loss fluid near the shroud wall instead of letting it convect down the suction side. However, the flow in the near-shroud region was backward, and the slot generated increased
Figure 4.10  Geometry fc: meridional view
Figure 4.11 Calculation fc: $P^*$ on cross-sectional planes; near-shroud velocity vectors
losers.

The geometry fd, shown in Fig. 4.12, had five 15 percent holes in the shroud region. The holes channeled the tip leakage flow and prevented the large backflow seen in calculation fc. The tip leakage flow, however, mixed out rapidly, and did not cause flow from the suction to the pressure side of the passage across the shroud (Fig. 4.13). Inertia caused secondary flows down the suction surface of the blade, and the high loss fluid was moved to the hub/suction-side corner region at the impeller exit. The net effect of the holes was a stratified blade-to-blade flow similar to the baseline calculation but with lower overall performance.

Overall, tip slots and holes increased the amount of losses and reduced the head rise, as seen below. The only case, fc, that reduced the flow distortion had a substantial reduction in head rise and efficiency.

<table>
<thead>
<tr>
<th></th>
<th>$\delta P_t/2$</th>
<th>$-\delta P^*$</th>
<th>$(a_{\text{max}}-a_{\text{min}})_c$ diff. inlet</th>
</tr>
</thead>
<tbody>
<tr>
<td>baseline</td>
<td>0.611</td>
<td>0.027</td>
<td>1.99°</td>
</tr>
<tr>
<td>fa</td>
<td>0.594</td>
<td>0.046</td>
<td>2.56°</td>
</tr>
<tr>
<td>fb</td>
<td>0.596</td>
<td>0.030</td>
<td>2.20°</td>
</tr>
<tr>
<td>fc</td>
<td>0.549</td>
<td>0.083</td>
<td>1.66°</td>
</tr>
<tr>
<td>fd</td>
<td>0.603</td>
<td>0.037</td>
<td>2.17°</td>
</tr>
</tbody>
</table>

Goto was able to achieve an improvement in exit flow distortion by using small tip slots because the shroud inlet boundary layer for his impeller was 25
Figure 4.12 Geometry fd: meridional view
Figure 4.13 Calculation fd: $P^*$ on cross-sectional planes; near-shroud velocity vectors
to 30 percent of the passage height with an estimated displacement thickness of 2-3 percent. In the Consortium impeller, however, the effective inlet boundary layer on the shroud was quite large, 85 percent of the blade height (see Fig. 4.3); and the displacement thickness was about 22 percent, an order of magnitude larger than the displacement thickness of Goto's shroud boundary layer. Therefore, small tip gaps did not succeed in convecting a significant portion of the shroud boundary layer in the Consortium impeller, whereas large slots and holes significantly increased the overall losses.
4.3.3 Blade Lean

Geometries la and lb

The baseline impeller was modified by leaning the blades in an attempt to reduce the circumferential distortion of the diffuser inlet flow. Several geometries were created using the blade lean concept. In the first two geometries, la and lb, the blade was modified by changing the distribution of the tangential grid coordinate after the splitter blade leading edge. The modification left the mid-height profiles unchanged and added a change in the tangential coordinate equal to:

$$\delta \theta = \text{constant} \ (A-1.13)^{1.6}(C-5)$$ [4.3.1]

As a result of the modification, the discharge blade angle distribution, which was constant (38°) in the baseline geometry, changed to a nearly linear distribution, with, for example, a 46° angle at the hub and a 32° at the tip for la. Geometry la had a backward lean, with an exit blade lean angle of 62°, while geometry lb had a forward lean of the same magnitude. Figure 4.14 shows the hub to shroud change in $\theta$ versus the non-dimensional distance parameter A for geometries la and lb. A three-dimensional view of the impeller geometry la is shown in Fig. 4.15.

The $p$, and $p^*$ contours that resulted from calculation la are shown in Fig. 4.16. The reduced static pressure contours are essentially unchanged from the
Figure 4.14 Blade lean for the baseline, la and lb geometries. $dth = \theta_{shroud} - \theta_{hub}$ along quasi-orthogonals.
Figure 4.16 Calculation la: $P_*$ and $P^*$ on cross-sectional planes.
baseline calculation, see Fig 4.5. However, since the blades are leaned backward, they are positioned differently relative to the \( p_c \) contours. Toward the impeller exit, in particular, the lowest \( p_c \) region is in the hub/suction-side corner.

The overall loss levels of the calculation la are the same as for the baseline impeller. Since the lowest \( p_c \) region has moved to the hub/suction-side corner, though, the high loss fluid is driven by secondary flows to the suction side and towards the hub corner. The inertia of the enhanced secondary flows causes the high loss fluid to spread across the hub at the exit. The end result is an exit flow that is stratified hub-to-shroud rather than blade-to-blade.

The near-blade and near-wall velocity vector plots are shown in Fig. 4.17 and Fig. 4.18, respectively. The secondary flows down the suction side of both the full blade and the splitter blade are stronger than in the baseline impeller, see Fig. 4.6. In addition, there is less stagnation of the flow in the hub/pressure-side corner because the reduced pressure is lower and the corner is obtuse. There is no backflow with the exception of the blades’ leading edges. The secondary flows from the suction side to the pressure side of the passage near the exit at the impeller hub are stronger than in the baseline calculation. In addition, the velocity vectors at the impeller exit appear more uniform than in the baseline impeller.

The diffuser inlet flow distortion is shown in Fig. 4.19. The diffuser inlet
Figure 4.17 Calculation Ia: near-blade velocity vectors
Figure 4.18 Calculation la: near-hub and near-shroud velocity vectors; velocity vectors for all heights at the impeller exit showing the exit flow angles.
Absolute velocity squared, $\delta C^2 = 0.025$

Tangent of absolute flow angle, $\delta Cr/C\theta = 0.01$

Figure 4.19 Calculation 1a: diffuser inlet plane flow distortion
flow is more uniform than in the baseline calculation, see Fig. 4.9, particularly in the circumferential direction.

The forward lean in geometry lb caused the low reduced static pressure region to lay in the shroud/suction-side corner, as seen in Fig. 4.20. The rotary stagnation pressure is stratified blade-to-blade at the exit, with the highest losses towards the shroud. Figure 4.21 shows the diffuser inlet flow distortion. From the absolute flow angle plot, it appears that there is a large axial variation, including a region of backflow near the shroud wall. Thus this geometry is not an improvement over the baseline.

The results for these initial tests with blade lean are summarized as follows:

<table>
<thead>
<tr>
<th></th>
<th>$\delta P_f/2$</th>
<th>$-\delta P^*$</th>
<th>$(\sigma_{\text{max}}-\sigma_{\text{min}})_c$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>diff. inlet</td>
</tr>
<tr>
<td>baseline</td>
<td>0.611</td>
<td>0.027</td>
<td>1.99°</td>
</tr>
<tr>
<td>la</td>
<td>0.621</td>
<td>0.027</td>
<td>1.15°</td>
</tr>
<tr>
<td>lb</td>
<td>0.603</td>
<td>0.028</td>
<td>1.31°</td>
</tr>
</tbody>
</table>

Geometry la has considerably less circumferential distortion than the baseline geometry, 1.15 degrees compared to 1.99 degrees, and it slightly improves the overall performance.
Absolute velocity squared, \( \delta C^2 = 0.025 \)

Tangent of absolute flow angle, \( \delta \frac{Cr}{C\theta} = 0.01 \)

Figure 4.21 Calculation lb: diffuser inlet plane flow distortion
Geometries Rk-a and Rk-b

Geometry la succeeded in reducing the circumferential distortion of the diffuser inlet flow. It was not completely clear, however, whether the improvement was due to the leaning of the blade or to the hub-to-shroud variation of the discharge blade angle which resulted from the application of blade lean.

In order to separate the effects of blade lean and hub-to-tip discharge blade angle variation, Rocketdyne created two geometries, Rk-a and Rk-b. In geometry Rk-a, the discharge blade angle varied as in la and there was no blade lean, while geometry Rk-b had constant discharge blade angle and 45° backward lean. Figure 4.22 shows the distributions of the blade lean for the two Rocketdyne geometries compared to the lean of the baseline and la geometries.

The pressure distributions for the impellers Rk-a and Rk-b are shown in Figs. 4.23 and 4.24, respectively. The $P_*$ and $P^*$ contours of calculation Rk-a appear similar to the results of the baseline calculation (Fig. 4.5). The results of calculation Rk-b, on the other hand, show rotary stagnation pressure with hub-to-shroud variation at the impeller exit similar to calculation la (Fig. 4.16). The results of calculations Rk-a and Rk-b,
Figure 4.22  Blade lean: geometries la, Rk-a, and Rk-b. $d\theta = \theta_{shroud} - \theta_{hub}$ along quasi-orthogonals.
Figure 4.23 Calculation Rk-a: Pr and P* on cross-sectional planes
<table>
<thead>
<tr>
<th></th>
<th>$\delta P_\ell/2$</th>
<th>$-\delta P^*$</th>
<th>$\left(\alpha_{\text{max}}-\alpha_{\text{min}}\right)_c$ diff. inlet</th>
</tr>
</thead>
<tbody>
<tr>
<td>baseline</td>
<td>0.611</td>
<td>0.027</td>
<td>1.99°</td>
</tr>
<tr>
<td>Rk-a</td>
<td>0.612</td>
<td>0.027</td>
<td>2.33°</td>
</tr>
<tr>
<td>Rk-b</td>
<td>0.614</td>
<td>0.028</td>
<td>1.76°</td>
</tr>
</tbody>
</table>

confirm that the flow improvements obtained by the geometry la are due to the blade lean, and not to the variation in the exit blade angle. But even though the geometry Rk-b shows some improvement over the baseline geometry in terms of circumferential distortion, 1.76 degrees compared to 1.99 degrees, it is not as good as the geometry la which gave 1.15 degrees.
Geometries lc and ld

Although geometry la was successful in reducing the diffuser inlet flow distortion, the drastic 62° backward lean would likely prove difficult to manufacture and could create high stress concentrations in the corner regions. Velocity distortion parameters for the leaned geometries were plotted as a function of the blade lean angle at both the impeller exit and the diffuser inlet planes (Figs. 4.25 and 4.26). The data indicate that both forward and backward blade lean improve circumferential distortion. The axial distortion and the overall velocity distortion parameters, however, are worse with forward lean at the diffuser inlet plane.

Based on the geometry Rk-b, 45° was determined to be a safe blade lean limit. In addition, the data suggest that a redesigned geometry with a 45° exit blade lean could have very low distortion. Therefore, two 45° exit backward lean geometries were created.

Geometry lc was generated by averaging the tangential grid coordinates of la and the baseline grid in an approximately 3 to 2 ratio. The resulting geometry had a blade lean angle of 45° at the exit, equal to the blade lean in geometry Rk-b, but with a different blade shape. The procedure that was used to design geometry lc is illustrated in Fig. 4.27. The results of the MEFP calculation show an impeller exit pressure distribution, shown in Fig. 4.28, that is similar to the calculation la (Fig. 4.16). Figure 4.29 presents the flow
Figure 4.25 Velocity distortion parameters vs. blade lean at impeller exit
NOTE: 0 = baseline, A = !a, B = !b, a = Rk-a, b = Rk-b
Figure 4.26  Velocity distortion parameters vs. blade lean at diffuser inlet
NOTE: O = baseline, A = Ia, B = Ib, a = Rk-a, b = Rk-b
\[ \theta_{lc} = \theta_{baseline} + x \cdot (\theta_{la} - \theta_{baseline}) = x \cdot \theta_{la} + (1.0 - x) \theta_{baseline} \]

\[ x = 0.58577 \]

Figure 4.27 Determination of geometry lc
Figure 4.28 Calculation of $P_v$ and $P_*$ on cross-sectional planes
Absolute velocity squared, $\delta C^2 = 0.025$

Tangent of absolute flow angle, $\delta Cr/C\theta = 0.01$

Figure 4.29 Calculation lc: diffuser inlet plane flow distortion
distortion at the diffuser inlet plane, which also appears similar to that of
calculation la (Fig. 4.19).

Finally, another 45° blade lean grid, ld, was created by averaging the
tangential coordinates of geometries Rk-b and lc. Again, the P, contours (Fig.
4.30) remained fixed in space compared to the baseline calculation (Fig. 4.5),
but the leaning of the blade changed the location of the minimum value of P,
and the loss distribution became hub-to-tip. The results for these cases are
summarized as follows:

|       | $\delta P^*/2$ | $-\delta P^*$ | $(\alpha_{max} - \alpha_{min})_c$
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>baseline</td>
<td>0.611</td>
<td>0.027</td>
<td>1.99°</td>
</tr>
<tr>
<td>lc</td>
<td>0.616</td>
<td>0.027</td>
<td>1.66°</td>
</tr>
<tr>
<td>ld</td>
<td>0.616</td>
<td>0.027</td>
<td>1.56°</td>
</tr>
</tbody>
</table>

Both geometries lc and ld had improved circumferential distortion parameters
relative to the baseline geometry, 1.66 and 1.56 degrees compared to 1.99
degrees, but again they were not as good as geometry la, 1.15 degrees.
Figure 4.30 Calculation for $P_r$ and $P^*$ on cross-sectional planes
4.3.4 Flow Distortion

All three impeller exit flow distortion parameters from section 2.5 were calculated for the baseline geometry as well as for all the blade lean geometries. Table 4.1 shows the velocity distortion parameters at the diffuser inlet plane, while Table 4.2 presents the distortion parameters at the impeller exit plane.

It can be observed that both the distortion in the circumferential direction, \( (\alpha_{\text{max}} - \alpha_{\text{min}})_c \), and the overall distortion parameter, \( B_t \), decay quickly in the vaneless region. The distortion in the axial direction, \( (\alpha_{\text{max}} - \alpha_{\text{min}})_{\text{ha}} \), on the other hand, either increases or decreases depending on the geometry. These observations are consistent with the experimental results obtained by both Yoshinaga et al. (11) and Agrawal et al. (12).

In this first phase of the Consortium impeller redesign, the main objective was optimizing the circumferential distortion without loss in head or efficiency. Based on this, of the buildable geometries (with blade lean not higher than 45 degrees) Id has the lowest circumferential distortion, followed closely by Ic. Considering the other distortion parameters as well, \( (\alpha_{\text{max}} - \alpha_{\text{min}})_{\text{ha}} \) and \( B_t \), the optimum impeller is a choice between Ic, with the lowest \( B_t \) and \( (\alpha_{\text{max}} - \alpha_{\text{min}})_{\text{ha}} \), and Id, with the lowest \( (\alpha_{\text{max}} - \alpha_{\text{min}})_c \). If the distortion parameters were evaluated at the impeller exit, of the buildable geometries, Rk-b has the lowest overall and circumferential distortions, but it has the highest axial distortion of any
calculation. Lc and Id are reasonable compromises with relatively low values for all three distortion parameters.

With the exception of the geometries with slots or holes, all the designs have quite similar head rise and efficiency values. Thus, if the impeller design choice had been made at this stage of the redesign process, the recommendation would have been either geometry lc or Id.
Table 4.1 Velocity distortion parameters at diffuser inlet

<table>
<thead>
<tr>
<th></th>
<th>$B_t$</th>
<th>$(\alpha_{min} - \alpha_{min})_{ha}$</th>
<th>$(\alpha_{min} - \alpha_{min})_{c}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>baseline</td>
<td>1.03</td>
<td>6.54°</td>
<td>1.99°</td>
</tr>
<tr>
<td>fa</td>
<td>1.05</td>
<td>6.62°</td>
<td>2.56°</td>
</tr>
<tr>
<td>fb</td>
<td>1.05</td>
<td>8.34°</td>
<td>2.20°</td>
</tr>
<tr>
<td>fc</td>
<td>1.09</td>
<td>10.69°</td>
<td>1.55°</td>
</tr>
<tr>
<td>fd</td>
<td>1.04</td>
<td>5.82°</td>
<td>2.17°</td>
</tr>
<tr>
<td>la</td>
<td>1.02</td>
<td>5.58°</td>
<td>1.15°</td>
</tr>
<tr>
<td>lb</td>
<td>1.09</td>
<td>10.17°</td>
<td>1.31°</td>
</tr>
<tr>
<td>Rk-a</td>
<td>1.05</td>
<td>5.69°</td>
<td>2.33°</td>
</tr>
<tr>
<td>Rk-b</td>
<td>1.08</td>
<td>10.04°</td>
<td>1.76°</td>
</tr>
<tr>
<td>lc</td>
<td>1.02</td>
<td>5.50°</td>
<td>1.66°</td>
</tr>
<tr>
<td>ld</td>
<td>1.03</td>
<td>5.89°</td>
<td>1.56°</td>
</tr>
</tbody>
</table>
Table 4.2  Velocity distortion parameters at impeller exit

<table>
<thead>
<tr>
<th></th>
<th>$B_l$</th>
<th>$(\alpha_{\text{max}} - \alpha_{\text{min}})_{\text{ha}}$</th>
<th>$(\alpha_{\text{max}} - \alpha_{\text{min}})_{\text{c}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>baseline</td>
<td>1.53</td>
<td>4.82°</td>
<td>16.63°</td>
</tr>
<tr>
<td>fa</td>
<td>1.65</td>
<td>3.68°</td>
<td>18.97°</td>
</tr>
<tr>
<td>fb</td>
<td>1.64</td>
<td>9.35°</td>
<td>18.94°</td>
</tr>
<tr>
<td>fc</td>
<td>1.74</td>
<td>7.73°</td>
<td>19.22°</td>
</tr>
<tr>
<td>fd</td>
<td>1.52</td>
<td>5.16°</td>
<td>16.51°</td>
</tr>
<tr>
<td>la</td>
<td>1.38</td>
<td>7.87°</td>
<td>6.43°</td>
</tr>
<tr>
<td>lb</td>
<td>1.59</td>
<td>3.12°</td>
<td>7.26°</td>
</tr>
<tr>
<td>Rk-a</td>
<td>1.55</td>
<td>3.14°</td>
<td>19.74°</td>
</tr>
<tr>
<td>Rk-b</td>
<td>1.41</td>
<td>10.44°</td>
<td>8.86°</td>
</tr>
<tr>
<td>lc</td>
<td>1.56</td>
<td>7.35°</td>
<td>9.90°</td>
</tr>
<tr>
<td>ld</td>
<td>1.52</td>
<td>8.66°</td>
<td>9.08°</td>
</tr>
</tbody>
</table>
4.4 Second Phase of Impeller Redesign - High Head Coefficient

After applying CFD techniques to the design of the baseline impeller, it appeared to NASA Marshall and Rocketdyne that it could be possible to obtain a higher head coefficient from an impeller of the same diameter.

As a result, during the second phase of the Consortium impeller redesign, the emphasis changed from the reduction of the exit flow distortion to the achievement of a higher head coefficient. Specifically, the head coefficient requirement was raised from 0.6 to 0.7.

Impeller Geometries

Four different geometries were created by Rocketdyne in a design process that concluded with the selection of the final impeller design. The new geometries are labeled HH 1, HH 3, HH 4, and HH 5. The high head geometries, also referred to as Advanced Concept Impeller geometries, have the same inlet duct and the same exit diameter as the baseline impeller, but have a different meridional profile. Figure 4.31 shows the meridional view comparison of the high head geometries and the baseline geometry. The high head impellers have a longer inlet section and a smaller impeller exit height. The Advanced Concept Impeller geometries have the same rotational speed and flow rate as the baseline impeller, but with higher heads they have lower specific speeds. The high head geometries also have six full blades and six
Figure 4.31 Meridional view comparison: baseline and high head geometries, see also Fig. 4.2.
splitter blades as in the baseline design.

The four high head geometries differ in the meridional location of the splitter blade leading edge, which can be seen in Fig. 4.32. In addition, the blade wrap and the incidence of the full and splitter blade leading edges vary from geometry to geometry. Figure 4.33 compares the blade profiles at the shroud of the high head geometries with the baseline geometry.

The full blade wrap angles at the shroud (FBWA) are as follows:

<table>
<thead>
<tr>
<th></th>
<th>baseline</th>
<th>HH 1</th>
<th>HH 3</th>
<th>HH 4</th>
<th>HH 5</th>
</tr>
</thead>
<tbody>
<tr>
<td>FBWA</td>
<td>79°</td>
<td>57°</td>
<td>60°</td>
<td>69°</td>
<td>69°</td>
</tr>
</tbody>
</table>

**Pressures and Velocity Distributions**

The mass averaged pressures of calculation HH 1 are shown in Fig. 4.34. The head coefficient of the impeller is 0.681, quite close to the target value of 0.7. Virtually all the head rise, however, is achieved after the splitter blade leading edge, and the overall losses are quite high compared with the losses in the baseline calculation; $-\delta P^*$ increases from 0.027 to 0.064, and the efficiency falls from 97.8 percent to 95.5 percent.

The reduced static pressure and rotary stagnation pressure contours are shown in Fig. 4.35. The losses are higher than the baseline losses (Fig. 4.5) from the inlet, and continue to be higher throughout the passage. At the impeller exit, the losses are stratified blade to blade. Also, it may be noticed
Figure 4.33 HH1 geometries: blade profile comparison at the shroud

NOTE: the shaded sections show the changed geometry

- HH 3
- HH 4
- HH 5
- HH 1
- HH 3
- HH 1
- Baseline
Mass-averaged pressures
t-total, s-static, *-rotary stagnation, r-reduced

\[ \delta P_t = 1.557 - 0.195 = 1.362 \]
\[ \delta P^* = -0.090 - (-0.026) = -0.064 \]
Eff = 95%
Figure 4.35: Calculation HH 1; P, and P* on cross-sectional planes
in Fig. 4.35 that there is a small amount of blade lean throughout the impeller, but not as much near the blade inlet as in the baseline design, Fig. 4.5. At the impeller exit, the lean is 30 degrees.

The near wall velocity vectors are shown in Fig. 4.36. There is strong backflow upstream of the blades’ leading edges at the shroud. The backflow is caused by the attempt to turn the flow too quickly in the inlet section and instead it separates. The flow is quite non-uniform by the exit, and there is a visible jet/wake pattern, seen in Fig 4.36. The corresponding diffuser inlet plane distortion is shown in Fig. 4.37.

Geometry HH 3 was designed to maintain a high head coefficient while reducing the large backflow region across the shroud seen in calculation HH 1. The impeller has a smaller blade incidence, see Fig. 4.33, and the flow turning is delayed. This reduced the backflow across the shroud upstream of the leading edge as seen in Fig. 4.38. Geometry HH 4 was designed with similar objectives but with a larger amount of blade wrap, see Fig. 4.33. This allows the critical section of the suction side of the full blade where the flow is turned to be covered upstream by the next impeller blade. This design feature prevents backflow from propagating upstream (as seen in Fig. 4.39) and reflecting adversely upon the overall performance of the impeller.

Figure 4.39 also shows a near stagnation condition along part of the pressure side of geometry HH 4. This flow behavior is created as the impeller
Figure 4.36 Calculation HH 1: near-hub and near-shroud velocity vectors; velocity vectors for all heights at the impeller exit showing the exit flow angles.
Absolute velocity squared, $\delta C^2 = 0.025$

Tangent of absolute flow angle, $\delta C_r/C\theta = 0.01$

Figure 4.37 Calculation HH 1: diffuser inlet plane flow distortion
Figure 4.38 Calculation HH 3: near-hub and near-shroud velocity vectors; velocity vectors for all heights at the impeller exit showing the exit flow angles.
Figure 4.39 Calculation HH 4: near-hub and near-shroud velocity vectors; velocity vectors for all heights at the impeller exit showing the exit flow angles.
geometry is pushed to achieve a high head rise.

The rotary stagnation pressure contours of calculations HH 3 and HH 4 are shown in Fig. 4.40. Geometry HH 3 shows significantly reduced losses over HH 1 near the inlet (A=0.4), and for HH 4 the losses are even more reduced. Both geometries have more circumferentially uniform losses at the exit than geometry HH 1 (see Fig 4.35).

The loss distributions at the exit of impeller HH 4 are different for the two flow passages. Similarly, the flow split is quite uneven. 56 percent of the flow passes between the suction surface of the full blade and the splitter blade and 44 percent of the flow passes between the pressure surface of the full blade and the splitter blade. This contributes to the high exit flow distortion.

To improve the flow split, a final geometry, HH 5, was created. The only change made from HH 4 to HH 5 is that the splitter blade leading edge was moved 2.5° towards the full blade suction side, see Fig 4.33. This small geometry change proved to be beneficial to the impeller flows, and the flow split improved to 54/46.

All the results for this final geometry are shown in Figs. 4.41-4.44. The predicted overall performance seen in Fig. 4.41 is excellent, with a head coefficient of 0.696 and an efficiency of 98 percent. The pressure contours are shown in Fig. 4.42. The losses in the two passages have become more similar due to the improved flow split. Figures 4.43 and 4.44 show the near-
Figure 4.40 Calculations HH 3 and HH 4: $P_e$ on cross-sectional planes
Mass-averaged pressures
t-total, s-static, *-rotary stagnation, r-reduced

\[
\delta P_t = 1.587 \cdot .195 = 1.392 \\
\delta P^* = -.054 \cdot (-.026) = -.028 \\
\text{Eff} = 98 \%
\]

Figure 4.41 Calculation HH 5: mass-averaged pressures
Figure 4.42 Calculation HH 5; P, and P* on cross-sectional planes

A = 0.4

1.4

1.8

2.0

Pr

p

s
Figure 4.43 Calculation HH 5: near-blade velocity vectors
Figure 4.44 Calculation HH 5: near-hub and near-shroud velocity vectors; velocity vectors for all heights at the impeller exit showing the exit flow angles.
blade and near-wall velocity vectors, which are quite similar to those from calculation HH 4. The diffuser inlet flow distortion is given in Fig. 4.45.
Absolute velocity squared, $\delta C^2 = 0.025$

Tangent of absolute flow angle, $\delta \frac{Cr}{C\theta} = 0.01$

Figure 4.45 Calculation HH 5: diffuser inlet plane flow distortion
Overall Performance

Overall performance parameters for the Advanced Concept Impeller geometries are shown in Table 4.3. All the high head geometries achieved the desired head rise. However, the losses in calculations HH 1 and HH 3 are a factor of two higher than the baseline losses. These large losses and the backflow in HH 1 reduce the confidence in these results. Both geometries HH 4 and HH 5 have loss levels similar to the baseline, making these candidates for the final design choice.

Table 4.4 gives the distortion parameters for the high head geometries at the diffuser inlet and impeller exit planes. For most of the distortion parameters, there is little difference between geometries HH 4 and HH 5. However, HH 5 has a significantly lower $B_r$ at the diffuser inlet plane than HH 4. The results of Agrawal et al. (Fig 2.13) suggest that a $B_r$ of 1.15 for HH 5 compared to 1.22 for HH 4 would significantly improve the performance of a vaned diffuser.
Table 4.3 High head geometries: overall performance parameters.

<table>
<thead>
<tr>
<th></th>
<th>head coefficient</th>
<th>losses/baseline losses</th>
<th>flow split</th>
</tr>
</thead>
<tbody>
<tr>
<td>baseline</td>
<td>.611</td>
<td>1.0</td>
<td>48/52</td>
</tr>
<tr>
<td>HH 1</td>
<td>.681</td>
<td>2.4</td>
<td>50/50</td>
</tr>
<tr>
<td>HH 3</td>
<td>.694</td>
<td>2.0</td>
<td>54/46</td>
</tr>
<tr>
<td>HH 4</td>
<td>.697</td>
<td>1.07</td>
<td>56/44</td>
</tr>
<tr>
<td>HH 5</td>
<td>.696</td>
<td>1.04</td>
<td>54/46</td>
</tr>
</tbody>
</table>
Table 4.4 High head geometries: velocity distortion parameters

At diffuser inlet

<table>
<thead>
<tr>
<th></th>
<th>$\alpha_{\text{max}} - \alpha_{\text{min}}$&lt;sub&gt;hs&lt;/sub&gt;</th>
<th>$\alpha_{\text{max}} - \alpha_{\text{min}}$&lt;sub&gt;c&lt;/sub&gt;</th>
<th>$B_i$</th>
</tr>
</thead>
<tbody>
<tr>
<td>baseline</td>
<td>6.54°</td>
<td>1.98°</td>
<td>1.03</td>
</tr>
<tr>
<td>HH 1</td>
<td>4.47°</td>
<td>4.09°</td>
<td>1.06</td>
</tr>
<tr>
<td>HH 3</td>
<td>15.45°</td>
<td>4.05°</td>
<td>1.05</td>
</tr>
<tr>
<td>HH 4</td>
<td>7.35°</td>
<td>2.70°</td>
<td>1.22</td>
</tr>
<tr>
<td>HH 5</td>
<td>5.53°</td>
<td>2.89°</td>
<td>1.15</td>
</tr>
</tbody>
</table>

At impeller exit

<table>
<thead>
<tr>
<th></th>
<th>$\alpha_{\text{max}} - \alpha_{\text{min}}$&lt;sub&gt;hs&lt;/sub&gt;</th>
<th>$\alpha_{\text{max}} - \alpha_{\text{min}}$&lt;sub&gt;c&lt;/sub&gt;</th>
<th>$B_i$</th>
</tr>
</thead>
<tbody>
<tr>
<td>baseline</td>
<td>4.82°</td>
<td>16.63°</td>
<td>1.53</td>
</tr>
<tr>
<td>HH 1</td>
<td>4.35°</td>
<td>17.89°</td>
<td>1.71</td>
</tr>
<tr>
<td>HH 3</td>
<td>6.10°</td>
<td>14.76°</td>
<td>1.63</td>
</tr>
<tr>
<td>HH 4</td>
<td>6.88°</td>
<td>11.94°</td>
<td>1.49</td>
</tr>
<tr>
<td>HH 5</td>
<td>7.41°</td>
<td>12.23°</td>
<td>1.49</td>
</tr>
</tbody>
</table>
Final Geometry Selection

Overall, geometry HH 5 has a high head coefficient, low losses, and acceptably low exit flow distortion. For these reasons, the VPI & SU team recommended geometry HH 5 as the best Advanced Concept Impeller geometry. In October 1993, NASA Marshall and Rocketdyne reviewed the results provided by the Pump Stage Team members as well as their own calculations, and chose HH 5 as the final geometry.

The impeller will be built and tested by Rocketdyne in early 1994. The experimental data will be made available to the PST members, and will be compared with the CFD results by NASA Marshall.
5.0 CONCLUSIONS

The redesign of the Consortium impeller in which the VPI & SU team participated and which is described in this thesis represents a significant study of current CFD capabilities in turbomachinery design. Several design modifications were made and the resulting flow changes were analyzed using the Moore Elliptic Flow Program (MEFP). The conclusions from calculations for fifteen impeller geometries are:

1. Tip slots or holes are not effective in reducing exit flow distortion when the impeller inlet boundary layer is large.

2. Exit flow distortion can be reduced by leaning the blades. Blade lean leaves the pressure distribution nearly unchanged in space. However, since the blades are leaned, they are positioned differently relative to the $P_r$ contours, and the location of the lowest $P_r$ region is changed. In the present work, backward lean was used to reduce the circumferential flow variations at the diffuser inlet, resulting in a reduction in the diffuser excitation parameter.

3. The overall flow distortion parameter, $B_r$, and the circumferential flow distortion parameter $(a_{max} - a_{min})_c$ decay rapidly in the vaneless region. The axial flow distortion parameter, $(a_{max} - a_{min})_{he}$, however, tends to remain relatively
constant.

4. The head coefficient of the impeller can possibly be increased without reducing the efficiency or introducing excessive flow distortion. In the present work, the head coefficient for impeller HH 5 was raised from 0.6 to 0.7, while maintaining an efficiency of 98 percent. The results of calculation HH 5 show an acceptable vaned diffuser inlet flow distortion, $B_r = 1.15$.

5. Flow separations and loss generation at the impeller inlet can be reduced by designing the impeller for zero flow incidence at the leading edge of the blades. Separations and losses at the impeller inlet can also be reduced by increasing the blade wrap.

6. A more uniform flow split in the impeller passage appears to reduce the diffuser inlet flow distortion. The flow split can be modified by changing the tangential location of the splitter blade leading edge and/or improving the flow incidence at the splitter blade leading edge.

CFD codes can be useful in turbomachinery design, particularly since they provide quick feedback and very detailed output. The strongly non-uniform grid spacing and relatively coarse grid used in MEFP calculations
allowed an 18 hour turn-around time for each calculation on an HP 720 workstation. Without the aid of CFD codes, the final Consortium impeller design would have probably been quite different.

During this computational study, impeller geometries with lower exit flow distortion than the baseline geometry were designed. In addition, it appears possible to design high head coefficient impellers using computational methods. The planned testing is necessary to verify the predictions of 3-D flow codes. The testing will also provide an indication of the capabilities of CFD codes, particularly MEFP, for use in impeller redesign.
BIBLIOGRAPHY


VITA

Alessandro Lupi was born in Genoa, Italy, in 1970. He is the son of Ettore and Bruna Lupi. After completing three years of high school in Genoa, he graduated from Ridgeway High School in Memphis, Tennessee as an exchange student. He obtained a Bachelor’s degree in Mechanical Engineering from Christian Brothers University in Memphis in May 1992. In August 1992 he began his studies in the Master’s program in Mechanical Engineering at Virginia Tech, and worked first as a teaching assistant, and than as a research assistant under the sponsorship of Rocketdyne.