A Numerical Investigation of Fluid Flow for Disk Pumping Applications

D. V. ROSCOE
R. C. BUGGELN
J. A. FOSTER
and
H. MCDONALD
Scientific Research Associates, Inc.
Glastonbury, CT 06033

ABSTRACT

This paper describes numerically calculated results obtained for a viscous driven enclosed cavity flow and comparison with the experiments of Daly and Nece (Ref. 4). The sensitive prediction of moment coefficient is chosen for comparison purposes. Some discussion of the impact of k-ε turbulence modeling is also included. A second configuration demonstrating the degree of geometric complexity which can be handled is also presented.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>a</td>
<td>Disk radius</td>
</tr>
<tr>
<td>b</td>
<td>Disk thickness</td>
</tr>
<tr>
<td>C</td>
<td>Tip clearance</td>
</tr>
<tr>
<td>C₁, C₂, Cu</td>
<td>Constants defined in text</td>
</tr>
<tr>
<td>Cp</td>
<td>Specific heat</td>
</tr>
<tr>
<td>Cm</td>
<td>Moment coefficient</td>
</tr>
<tr>
<td>D</td>
<td>Rate of strain tensor</td>
</tr>
<tr>
<td>h₀</td>
<td>Total enthalpy</td>
</tr>
<tr>
<td>k</td>
<td>Turbulent kinetic energy</td>
</tr>
<tr>
<td>Pr</td>
<td>Prandtl number</td>
</tr>
<tr>
<td>p</td>
<td>Static pressure</td>
</tr>
<tr>
<td>q</td>
<td>Heat flux</td>
</tr>
<tr>
<td>R₀</td>
<td>Rotational Reynolds number</td>
</tr>
<tr>
<td>RT</td>
<td>Turbulent Reynolds number</td>
</tr>
<tr>
<td>s</td>
<td>Disk spacing</td>
</tr>
<tr>
<td>t</td>
<td>Time</td>
</tr>
<tr>
<td>T</td>
<td>Temperature</td>
</tr>
<tr>
<td>V</td>
<td>Velocity</td>
</tr>
<tr>
<td>ε</td>
<td>Turbulence dissipation rate</td>
</tr>
<tr>
<td>κ</td>
<td>Thermal conductivity</td>
</tr>
<tr>
<td>ρ</td>
<td>Density</td>
</tr>
<tr>
<td>τ</td>
<td>Stress tensor</td>
</tr>
<tr>
<td>μ</td>
<td>Laminar viscosity</td>
</tr>
<tr>
<td>μₜ</td>
<td>Turbulent viscosity</td>
</tr>
</tbody>
</table>

INTRODUCTION

The nature of both axial and centrifugal turbomachines inevitably leads to situations in which rotating and non-rotating components are in close proximity. This paper deals specifically with disk cavities which can arise as an enclosed cavity without inflow or outflow, though more usually fluid is pumped through the cavity to provide a means of cooling engine components. The understanding and prediction of these secondary gas path flow fields is of importance in allowing designers to accurately estimate loss and cooling air bleed rates to maintain adequate component cooling while minimizing the impact of secondary gas flow on engine performance. The use of CFD techniques in conjunction with experimental testing serves as an aid in predicting the nature of these flow fields and can help in guiding redesign so as to minimize loss while improving cooling. The present effort applies a well exercised Navier-Stokes computer code to the disk pumping cavity flow field problem. The work follows that of Kim, Buggeln and McDonald (Ref. 1) who used an earlier version of the present code to analyze flow in rectangular and shaped disk pump cavities. Although no attempts were made to compare with data in Ref. 1, the results clearly showed the effect of disk geometry and rotation upon the resulting flow pattern. Other applications of Navier-Stokes procedures to the disk pumping cavity are due to Chow (Ref. 2) and Morse (Ref. 3). The form of the present effort is to extend the work initiated in Ref. 1 to more complex flow situations, higher flow resolution and to include comparisons with experimental data. The present work forms part of a larger program in which the aim is to develop a general computer code for the prediction of the flow fields in secondary gas paths. Here, results obtained in both the laminar and turbulent regimes for an
enclosed disk test case (Ref. 4) and a demonstration calculation for flow through a complex shaped centrifugal impeller backface cavity are described.

The enclosed disk case is part of the validation exercise while the impeller backface cavity calculation is a demonstration of the geometric generality of the code.

ANALYSIS

In the present approach, the time dependent, ensemble-averaged, Navier-Stokes equations are numerically solved for general nonorthogonal body-fitted coordinate systems. In general, the equations solved are the continuity equation, the momentum equations, the energy equation and, if necessary, the turbulence kinetic energy equation and the dissipation of turbulence kinetic energy. For the case described here, the momentum equation for the swirl component of velocity was also solved. The governing partial differential equations are formulated in conservation form by application of a Jacobian transformation to the equations in either cartesian or cylindrical polar coordinates. An outline of the transformation as well as the transformed system of equations have been presented in Ref. 5. The vector form of the equations is described briefly below.

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0
\] (1)

The vector momenta equations are

\[
\frac{\partial (\rho \vec{V})}{\partial t} + \nabla \cdot (\rho \vec{V} \vec{V}) = -\nabla \rho + \nabla \cdot \vec{T}
\] (2)

where \( \vec{T} \) is the stress tensor (molecular and Reynolds) written in the appropriate coordinate system.

The energy conservation equation is written as

\[
\frac{\partial (\rho h)}{\partial t} + \nabla \cdot (\rho \vec{V} h) = -\nabla \cdot \vec{q} + \nabla \cdot (\vec{T} \cdot \vec{V})
\] (3)

where in this case the stagnation enthalpy form of the energy equation is used. In the above equation the stress tensor is given by

\[
\vec{T} = \mu_{\text{eff}} (2D - \frac{2}{3} \nabla \nabla)
\] (4)

where the rate of strain tensor, \( D \), is given by

\[
D = \frac{1}{2} \left( (\nabla \vec{V}) + (\nabla \vec{V})^T \right)
\] (5)

The effective viscosity, \( \mu_{\text{eff}} \), is the sum of the molecular and the turbulent viscosities

\[
\mu_{\text{eff}} = \mu + \mu_T
\] (6)

The turbulent viscosity, \( \mu_T \), is obtained from either a mixing length turbulence model or from the solution of the two partial differential equations for the turbulent kinetic energy and the dissipation of turbulent kinetic energy previously mentioned. The heat flux vector, \( \vec{q} \), is given by

\[
\vec{q} = -\left( \kappa + \kappa_T \right) \vec{V} T
\] (7)

where \( \kappa \) and \( \kappa_T \) are the molecular and turbulent thermal conductivities, respectively. In the present analysis, \( \kappa \) and \( \kappa_T \) are obtained assuming molecular and turbulent Prandtl numbers \( Pr \) and \( Pr_T \), i.e.,

\[
\kappa = \frac{\mu_{\text{eff}} \rho}{Pr}
\] (8)

\[
\kappa_T = \frac{\mu_{\text{eff}} \rho}{Pr_T}
\] (9)

The turbulence model used was the transitional two-equation (k-\( \epsilon \)) model discussed by Jones and Launder (Ref. 6), which is a modified form of the k-\( \epsilon \) turbulence model originally developed by Harlow and Nakayama (Ref. 7). The modifications devised by Jones and Launder allow the k-\( \epsilon \) model to be utilized throughout the viscous sublayer without any wall function assumption. The present approach uses very high near wall resolution and solves the k-\( \epsilon \) and mean flow equations to the wall without resorting to wall functions. This represents a significant advantage since wall functions may be difficult to specify in the complex flows found in disk pump cavities.

Following the method of Jones and Launder (Ref. 6) the turbulence kinetic energy equation is written in the form:

\[
\frac{3}{\partial t} \left( \rho k \right) + \nabla \cdot (\rho \vec{V} k) = \nabla \cdot \left( \mu_T \nabla k \right)
\] + 2\( \mu_T \) \( (D:D) \) - \( k \epsilon \) \( [-2c_2^2 (\rho k^{1/2})^2] \)

while the turbulence dissipation rate equation is written as

\[
\frac{3}{\partial t} \left( \rho \epsilon \right) + \nabla \cdot (\rho \vec{V} \epsilon) = \nabla \cdot \left( \frac{\mu_T}{\sigma_c} \nabla \epsilon \right)
\] + \( C_2 \mu_T \) \( (D:D) \) - \( C_2 \rho \) \( \frac{c}{k} \) \( [+2c_2^2 \rho k^{1/2}] \)

The Prandtl-Kolmogorov relation defines the turbulent viscosity as:

\[
\mu_T = \frac{c}{\epsilon} \frac{\rho k^2}{c}
\] (12)
while again following Jones and Launder (Ref. 6), $c_k$ and $c_e$ are set to 1.0 and 1.3, respectively. The constant $C_1$ is set to 1.43 while $C_2$ and $C_3$ are functions of the local turbulence Reynolds number, $R_T$, having the forms

$$C_2 = 1.92 [1.0 - 0.3 \exp(-R_T^2)]$$ (13)

$$C_3 = 0.09 \exp[C/([1.0 + 0.02 R_T])]$$ (14)

and

$$R_T = \frac{\rho k^2}{\nu \epsilon}$$ (15)

The last terms, enclosed in brackets, in Eqs. (10) and (11) are required for modeling in the low Reynolds number region near the walls and allows the use of $c = 0$ as the wall boundary condition. More detailed discussion of these terms can be found in Ref. 6. Various forms of Eqs. (13) and (14) have also been discussed by a number of authors, each form attempting to account for the near wall effective viscosity by matching particular experimental data. The present work utilizes the functional forms suggested by Jones and Launder with some modification of constants due to swirl. Details are given in the Results section.

NUMERICAL PROCEDURE

The numerical procedure used to solve the governing equations is the consistently split linearized implicit (LBI) scheme of Briley and McDonald (Ref. 8). The method centers around the use of a formal linearization technique adapted for the integration of initial value problems. The governing equations are replaced by an implicit time difference approximation. Terms involving nonlinearity at the implicit time level are linearized by Taylor series expansion in time about the solution at the known level, and spatial difference approximations are introduced. The result is a system of multi-dimensional coupled linear difference equations for the dependent variables at the implicit time level.

To solve these difference equations, the Douglas-Gunn (Ref. 9) procedure for generating alternating direction implicit (ADI) schemes as perturbations of fundamental implicit difference schemes is introduced. This technique leads to systems of coupled linear difference equations having block-banded matrix structure which can be solved efficiently by standard block elimination methods.

The linearization technique permits the solution of coupled nonlinear equations in one space dimension by a one-step noniterative scheme. Since no iteration is required to compute the solution for a single time step and only moderate effort is required for solution of the implicit difference equations, the method is computationally efficient; this efficiency is retained for the multidimensional problems by using ADI techniques. The method is also economical in terms of computer storage, in its present form requiring only two time levels of storage for each dependent variable. Discussion of the approach is intentionally brief, therefore, for further details of the LBI scheme consult Refs. 8 and 10.

RESULTS

Enclosed Disk Cavity

The first set of results apply to the enclosed disk cavity. Daily and Neece (Ref. 4) reported the results of an extensive range of experimental tests for viscous driven flow in an enclosed cavity. These consisted of parametric variations of rotational Reynolds number and disk spacing for both plane and tapered disks. The specific cases used for comparison were chosen to cover the range of the data and include Reynolds numbers in both the laminar and turbulent regimes for two separate disk spacings.

Figure 1 shows the dimensions of the experimental equipment and the section considered for modeling purposes. Initially, a 50x50 grid was used to simulate the cavity, but excluding the disk tip/shroud gap area. In later calculations, and in all results presented here, the grid had 150 radial and 100 axial grid points. To ensure adequate resolution of wall shear layers, the first grid point off the wall was situated at a normalized distance of $10^{-6}$, giving at least one grid point within the sublayer for turbulent flows. As a consequence, it was not necessary to assume the existence of wall functions as boundary conditions. Instead, no-slip was applied as a boundary condition on all velocity components on each wall except the rotating disk. Here, the swirl component of velocity was set to the value dictated by the disk rotation speed. Zero normal derivative of pressure was used as the boundary condition on the continuity equation.

The first section of Table 1 shows results for a range of rotational Reynolds number for the $s/a = 0.0255$ case. The flows for this spacing are laminar. As such, they provide a very good test case for the numerical simulation since the results are not complicated by turbulence model assumptions. The columns of Table 1 show disk moment coefficients, $C_m$, where the tip moment coefficient is that due to the shear forces on the disk tip and the net disk moment coefficient is the entire disk moment coefficient less the tip moment coefficient. In the case of the tip results, both the numerically computed and theoretical results are shown followed by a comparison. The theoretically calculated tip moment coefficients were taken from Ref. 4 and were

$$3$$
originally obtained by assuming Couette flow in the
disktip/shroud gap. For the disk there are three
results columns followed by three comparisons. In
turn, these are numerically computed, empirical and
experimentally measured moment coefficients and
percentage discrepancies between numerical and
empirical, numerical and experimental, and experi-
mental and empirical results. Both the empirical and
experimental values were taken from Ref. 4. Results
displayed in Table 1 demonstrate the code simulated
the viscous-driven flow correctly, producing realistic
flow conditions and torque coefficients to within a
maximum difference of 15% from empirically derived
coefficients. It should be noted that in deriving
these empirical relations, Daily and Nece have
determined four flow regimes and provided a curve fit
of torque coefficient versus \(R_d\) for each regime
separately. Thus, the accuracy of an empirically
derived coefficient near the edge of a flow regime may
be suspect. This possible source of error, as well as
general measurement error, and the inaccuracy of the
disk-tip calculation must be considered when comparing
these numbers. In regard to comparison between the
numerically computed and measured values, the
difference was within 92%. These comparisons between
modeled and theoretical tip moment coefficients show
very good agreement.

The second section of Table 1 shows an analogous
set of results for disk spacing of 0.115. Again the
flow is laminar and agreement between modeled and
observed results are within 10% except for the
\(R_d = 3.27 \times 10^3\) case which shows a relatively large
discrepancy of 18.7%. However, the experimental data
at this Reynolds number is subject to scatter with
values varying from 0.0137 to 0.0206 and the
numerically computed value clearly falls in this
range. In assessing these laminar flow results
presented in Tables 1 and 2, it should be noted that
moment coefficients represent a very sensitive item
for comparison. Small changes in skin friction
coefficient at the outer edge of the disk will result in
major changes in moment coefficient.

Having obtained these laminar results, consider-
ation was given to the turbulent flow regime. As a
first step, a simple mixing length model was used to
determine turbulent viscosity. Once the solution was
converted for the mixing length model, a calculation
was initiated from this converged solution for a k-c
model. Use of the basic k-c model of Jones and
Launder (Ref. 6) to determine turbulent viscosity
overpredicted the magnitude of the torque
coefficients. The unmodified k-c turbulence model
contains inherent problems in the near-wall region,
particularly an apparent over prediction of turbulent
viscosity and consequently shear-stress. A literature
search of other investigators using two-equation
models for similar problems (Refs. 2, 3, 11) showed
that the basic Jones-Launder model does not give
consistently good agreement for these types of
swirling flows.

Since the original Jones-Launder model did not
give good prediction of disk torque coefficient, a
modified model was sought. In considering possible
modifications to this basic model, guidance was sought
from existing models appearing in the literature.
Most modified k-c models focus attention on the near
wall damping of the viscosity field by modifying the
form of the model for \(C_v\) appearing in:

\[ \nu_t = C_v \rho k^2/\varepsilon \]

The form of the model most frequently used is:

\[ C_v = 0.09 \exp \left[ CR/(1.0 + 0.02 R_T) \right] \]

which follows closely the form of Jones and Launder
who chose \( CR = 2.5 \). In addition, the value of \( CR \)
is most often determined by trial and error matching
of experimental data. The indications are that no
universal value of \( CR \) nor any modified form of the
expression for \( C_v \) have been developed to date that can
yield acceptable predictions for a wide range of flow
situations. Therefore, under the present study the
value of \( CR \) was varied to see if a reasonable
agreement with data could be obtained for these
rotating disk cavity flows over a wide range of flow
conditions for a single value of \( CR \).

The results of the study showed that using the
Jones-Launder model while increasing the value of \( CR \)
to 8.0 produced good agreement with data over a wide
range of flow parameters. Table 2 shows torque
coefficients calculated from the turbulence model
results using the altered form of \( C_v \) calculation, for
both of the disk-to-endwall separation distances.
From Table 2 it is seen that the present code
predicted moment coefficients to within a maximum
difference of 10% of empirically-derived values, and
within 14% when compared to experimentally-measured
values.

It should be noted that the use of \( CR \) equal to
8.0 should not be construed to be a "new and improved"
model or an "optimum" value, but merely a value chosen
by trial and error to produce reasonable agreement with
experiments over a wide range of flow conditions for
this particular device. More importantly in view of
the very good agreement with the experimental data in
the laminar regime it can be stated that discrepancies
between prediction and observation in the turbulent
regime are most likely due to turbulence model
problems and not numerical or algorithmic
deficiencies. The departure of \( CR \) from previously
used values signifies a need for more detailed experimental data for flows of this type if predictive capabilities are to improve. It can be seen that with some fitting of the data, as performed in the current study, the current k-ε models can be used to provide additional data at other Rotational Reynolds' numbers and disk spacings for which no experimental data exists, provided these fall within the bounds for which some validation has taken place.

Results obtained using both the 50x50 and the 150x100 grid were essentially the same. Run times for the turbulent flow calculations using the k-ε model were 1.03x10^-5 sec/grid point/iteration, typically requiring 100 to 150 iterations to converge at each rotational Reynolds number, which for the 150,000 points used yields times of 325 to 490 CPU seconds on a Cray XMP.

Centrifugal Impeller Backface Cavity

As stated earlier, the purpose of the work reported here was the further development of an existing computer code for application to disk pumping applications and the validation of the resulting code. The enclosed disk results described previously form part of an ongoing effort to validate the code while the case described here is presented by way of demonstration of the general nature of the approach and, in particular, the ability to deal with complex geometric shapes. Figure 2 shows a sketch of the centrifugal impeller and the backface cavity. Fluid is extracted from the primary gas path of the impeller through an annular slot between the impeller exit and the vanless radial diffuser, as indicated in the figure. In this configuration, this fluid is intended to purge the cavity to help dissipate heat generated in the cavity. This is accomplished by venting the cavity as shown in Fig. 2. Added to this, the top wall of the cavity rotates, giving rise to swirling flow. It is clear from this description and the sketch that the secondary gas path is complex, with a number of rearward and forward facing steps. Fig. 3 shows the coordinate system used in calculating the impeller backface cavity flow field. This 220 radial by 110 axial grid is designed on the basis of previous experience to resolve the wall and free shear layers and flow around the corners.

Two calculations are reported. Each case considers the impeller backface geometry shown in Fig. 2, where the computational grid is shown in the r-z plane of a cylindrical polar coordinate system. The reference Reynolds number for each case was 7 x 10^6 and rotational speeds were zero and 10,000 rpm. The standard Jones and Launder (Ref. 6) two-equation k-ε turbulence model was applied to produce the results reported here. For reference purposes, Fig. 4 shows the flow near the entrance of the cavity for the zero rotation case where flow remains relatively parallel to the wall until influenced by the blockage of the at which point the flow is turned down and through the gap between the backstep and protrusion. While in Fig. 5, quite clearly the dominant feature is acceleration between the backface and protrusion where several recirculating flow regions are evident. Flow reversal on the top wall is insignificant.

For a rotational speed of 10,000 rpm major differences in the flow field can be seen. Figure 6 shows a comparable view of the inlet region for high rotational speed, as was shown in Fig. 4 for the zero rotation case. Clearly evident is the fact that a strong reverse flow exists along the cavity backface. Figure 7 shows a view comparable to that of Fig. 4 and graphically demonstrates the physics of the flow.

CONCLUDING REMARKS

A calculation procedure based upon solution of the Navier-Stokes equations has been developed for the disk pump cavity problem. The procedure is efficient and is capable of producing highly resolved flows economically. In turbulent flow, the procedure places at least one point within near wall sublayers, thus removing the need for wall function formulations. When applied to laminar flow in an enclosed disk, the calculated results show very good comparison with experimental data (Ref. 4). For turbulent cases it was found that application of the standard Jones and Launder (Ref. 6) k-ε turbulence model as well as modifications of this model proposed by various authors consistently overpredicted the wall shear stress and consequently the disk moment coefficient results. However, model parameter values could be found to give reasonably good results over a wide range of case parameters. In addition, the procedure was applied to the much more geometrically complex case of an impeller backface cavity. It should be noted that the latter case was actually performed prior to the calculations for the enclosed cavity and since no experimental data was available for comparison purposes, the cost of running the calculation was minimized by using a simple mixing length turbulence model. The purpose of the exercise was to develop and demonstrate basic capability to deal with geometrically complex configurations. As such, when the code has been validated against reliable experimental data for simple geometries, it can then be applied with some confidence to more complex cases.

ACKNOWLEDGEMENT

The authors gratefully acknowledge the support of this work under contract F33615-85-C-2599 awarded by Air Force Wright Aeronautical Laboratories, Propulsion Laboratory.
REFERENCES


<table>
<thead>
<tr>
<th>s/a = .0255</th>
<th>Tip</th>
<th>Net(Disk)</th>
<th>% difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>R_0</td>
<td>C_m (n)</td>
<td>C_m (TH)</td>
<td>%Δ</td>
</tr>
<tr>
<td>1.00x10^3</td>
<td>.1076</td>
<td>.1000</td>
<td>+5.6</td>
</tr>
<tr>
<td>4.48x10^3</td>
<td>.0239</td>
<td>.0226</td>
<td>+5.8</td>
</tr>
<tr>
<td>1.00x10^4</td>
<td>.0106</td>
<td>.0095</td>
<td>+11.6</td>
</tr>
<tr>
<td>4.48x10^4</td>
<td>.0023</td>
<td>.0023</td>
<td>0.0</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>s/a = .115</th>
<th>Tip</th>
<th>Net(Disk)</th>
<th>% difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>R_0</td>
<td>C_m (n)</td>
<td>C_m (TH)</td>
<td>%Δ</td>
</tr>
<tr>
<td>1.00x10^3</td>
<td>.1071</td>
<td>.1000</td>
<td>+7.1</td>
</tr>
<tr>
<td>4.48x10^3</td>
<td>.0238</td>
<td>.0220</td>
<td>+8.2</td>
</tr>
<tr>
<td>8.82x10^3</td>
<td>.0120</td>
<td>.0129</td>
<td>-7.0</td>
</tr>
<tr>
<td>3.27x10^4</td>
<td>.0032</td>
<td>.0030</td>
<td>+6.7</td>
</tr>
</tbody>
</table>

Table 1. Comparison of Calculated and Measured Disk Moment Coefficients for Laminar Flow.

- R_0 - rotational Reynolds number
- C_m - torque coefficient
- n - numerically calculated
- emp - empirically derived
- exp - experimentally measured
- th - theoretically calculated
- ' - disk tip
- N - net (total - disk tip)
### Table 2. Comparison of Calculated and Measured Disk Moment Coefficients for Turbulent Flow.

<table>
<thead>
<tr>
<th>$R_e$</th>
<th>$C_m(n)$</th>
<th>$C_m(emp)$</th>
<th>$C_m(exp)$</th>
<th>$%\Delta_1$</th>
<th>$%\Delta_2$</th>
<th>$%\Delta_3$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$3.12 \times 10^6$</td>
<td>0.0038</td>
<td>0.0035</td>
<td>0.0038</td>
<td>-8.6</td>
<td>0</td>
<td>-7.9</td>
</tr>
<tr>
<td>$4.48 \times 10^6$</td>
<td>0.0033</td>
<td>0.0032</td>
<td>0.0035</td>
<td>+3.1</td>
<td>-5.7</td>
<td>-8.6</td>
</tr>
<tr>
<td>$1.0 \times 10^7$</td>
<td>0.0025</td>
<td>0.0026</td>
<td>0.0029</td>
<td>-3.8</td>
<td>-13.8</td>
<td>-10.3</td>
</tr>
<tr>
<td>$5.0 \times 10^7$</td>
<td>0.0022</td>
<td>0.0020</td>
<td>N/A</td>
<td>+10.0</td>
<td>N/A</td>
<td>N/A</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>$R_e$</th>
<th>$C_m^N(n)$</th>
<th>$C_m^N(emp)$</th>
<th>$C_m^N(exp)$</th>
<th>$%\Delta_1^N$</th>
<th>$%\Delta_2^N$</th>
<th>$%\Delta_3^N$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$3.12 \times 10^6$</td>
<td>0.0044</td>
<td>0.0041</td>
<td>0.0042</td>
<td>+7.3</td>
<td>+4.8</td>
<td>-2.4</td>
</tr>
<tr>
<td>$4.48 \times 10^6$</td>
<td>0.0043</td>
<td>0.0038</td>
<td>0.0038</td>
<td>+13.2</td>
<td>+13.2</td>
<td>0</td>
</tr>
<tr>
<td>$1.0 \times 10^7$</td>
<td>0.0037</td>
<td>0.0033</td>
<td>0.0032</td>
<td>+12.1</td>
<td>+15.6</td>
<td>+3.1</td>
</tr>
<tr>
<td>$5.0 \times 10^7$</td>
<td>0.0028</td>
<td>0.0024</td>
<td>N/A</td>
<td>+16.7</td>
<td>N/A</td>
<td>N/A</td>
</tr>
</tbody>
</table>

Table 2. Comparison of Calculated and Measured Disk Moment Coefficients for Turbulent Flow.

- $R_e$ - Rotational Reynolds number
- $C_m$ - Torque coefficient
- $n$ - Numerically calculated
- $emp$ - Empirically derived
- $exp$ - Experimentally measured
- $N$ - Net (total - disk tip)

![Computational Model of Experimental Rig](image-url)
Figure 2. Sketch of an R-Z Section of the Impeller Backface Cavity shown in Relation to the Impeller.

Figure 3. Coordinate System for Centrifugal Impeller Backface Cavity.
Figure 4. Velocity Vectors Near Inflow for Zero Rotation Case.

Figure 5. Velocity Vectors Near Protruberance for Zero Rotation Case.
Figure 6. Velocity Vectors Near Inflow for 10,000 rpm Case.

Figure 7. Velocity Vectors Near Protruberance for 10,000 rpm Case.