A CFD ANALYSIS OF THE FLOW IN THE IMPELLER REAR CAVITY OF AEROENGINES

X. Liu and K. V. Patel
Pratt & Whitney Canada Incorporated
Mississauga, Ontario, Canada

ABSTRACT

A 3-D Navier-Stokes CFD code has been applied to analyze the complicated viscous driven flow in the centrifugal compressor impeller rear cavity of aeroengines. The calculation is compared with measurements. Three different configurations are analyzed. For each case the computed flow details provide valuable information for optimizing the engine design.

NOMENCLATURE

- \( C_w = \) mass flow parameter = \( Q/(\nu r_{tip}) \)
- \( C_{\mu} = \) constant in \( u_1 \) equation = 0.09
- \( C_{\varepsilon} = \) constants in \( c \) equation = 1.44
- \( C_{\tau_2} = \) constants in \( c \) equation = 1.92
- \( \rho_0 = \) total enthalpy
- \( k = \) turbulent kinetic energy
- \( K = \) K-factor = \( u_{flow}/U_{disk} \)
- \( p = \) pressure
- \( q = \) heat flux vector
- \( Q = \) volumetric flow rate
- \( r = \) radius, radial coordinate
- \( r_{tip} = \) impeller tip radius
- \( R_{rot} = \) rotational Reynolds number = \( \omega r_{tip}^2/\nu \)
- \( u_\theta = \) tangential velocity component
- \( \vec{V} = \) velocity vector
- \( u_\phi = \) relative tangential velocity
- \( x = \) axial coordinate
- \( \nu = \) kinematic viscosity
- \( \nu_1 = \) turbulent kinematic viscosity
- \( \rho = \) air density
- \( \varepsilon = \) turbulent energy dissipation rate
- \( \tau = \) shear stress tensor
- \( \omega = \) rotational speed of impeller

1. INTRODUCTION

Among the internal geometries in a small gas turbine for aeroengine application, the rear cavity of the centrifugal compressor impeller is subjected to complicated flow pattern. The impeller rotates at a high speed, ranging from 20000 to 50000 rpm. Of necessity, there is a clearance between the rotating impeller tip and the stationary diffuser casing. Air flow passes across this clearance and in many cases is used to provide air for turbine disk cooling and bearing chamber pressurization. As is shown in Fig.1, from the impeller tip, flow enters the impeller rear cavity. The left side (impeller rear side) of the cavity is rotating around the \( x \)-axis, and the other sides (combustor casing and bearing housing) of the cavity are stationary. The flow in the cavity is dominated by strong secondary flows with complicated vortex patterns. The flow is viscous, turbulent and axisymmetric with strong tangential swirling. The swirling distribution and the vortex pattern determine the temperature rise due to the windage (friction work), which in turn determines the thermal stress and durability of the impeller. The cavity pressure distribution determines axial load on the thrust bearing supporting the rotating assembly. The flow venting this cavity is extracted from the mainstream gas path, and therefore, affects the thermodynamic efficiency of the engine. Clearly, to provide an optimum design, a good understanding of the flow in the impeller rear cavity is important.

Detailed flow measurements within the impeller rear cavity in the engine operating environment is difficult and expensive due to the limited access to the complicated geometry, especially in typical turbofan engines. Therefore, during preliminary design, empirical methods based on the classic free rotating disk theory (or disk friction theory) may be used, for example the method of Newman(1983), Owen(1988) and Chew(1989). Such methods generally do not predict enough flow details and often have the assumption that a substantial inviscid core region ex-
continuity : \( \nabla \cdot (\rho \vec{V}) = 0 \) \hspace{1cm} (1)

momentum : \( \nabla \cdot (\rho \vec{V} \vec{V}) = -\nabla p + \nabla \cdot \tau \) \hspace{1cm} (2)

energy : \( \nabla \cdot (\rho V_h \vec{V}) = -\nabla q + \nabla (\vec{V} \cdot \tau) \) \hspace{1cm} (3)

The code solves the 3-D Reynolds-averaged Navier-Stokes equations with the \( k-e \) turbulence model according to Launder and Spalding (1974):

\[
\nu_t = C_{\mu} k^2/\epsilon
\] \hspace{1cm} (4)

The turbulent kinetic energy \( k \) and turbulent energy dissipation rate \( \epsilon \) are evaluated from the following transportation equations

\[
k \cdot \nabla \cdot (\rho V^2 k) = \text{Production} - \rho e + \text{Diffusion}
\] \hspace{1cm} (5)

\[
\epsilon \cdot \nabla \cdot (\rho V^2 \epsilon) = (C_1 \text{Production} - C_2 \rho e) \epsilon/k + \text{Diffusion}
\] \hspace{1cm} (6)

The standard log-law wall friction is used in the near wall region. The rotational Reynolds number for the current problem is \( 1.6 \times 10^7 \) which is high enough for the validity of the turbulence model outside the near wall region.

The governing equations (1) to (6) are solved by a finite-volume method on general non-orthogonal boundary-fitted grids. More details about this code can be found in the reference of Thomas et al. (1989). This code has been applied to a simple rotating disk cavity in Ref.15, and good agreement was obtained with the experimental results of Daily and Nece (1990). In the present calculation a 150 x 65 x 3 grid was generated for each configuration with CATIA software. Since the flow is axisymmetric, three meridian planes were used for the calculation. The grid is approximately orthogonal at the boundary, and inside the domain the deviation from orthogonal grid is less than 45 degrees. Relatively fine grids are used close the solid walls. The calculation was performed on an IBM-6000 work station. Each iteration step takes about 300 seconds CPU time. Typically after 110 iterations the maximum normalized residual of all equations reduces to below \( 1.0 \times 10^{-3} \) which is the normal convergence criterion. However the calculation was continued until the maximum normalized residual was reduced to below \( 0.4 \times 10^{-3} \), typically after 140 iterations. A comparison between the results after 140 iterations and after 110 iterations showed no significant difference. Hence good convergence has been achieved.

3. RESULTS AND DISCUSSIONS

Three different configurations are investigated in order to understand the various effects on the impeller rear cavity flow, so that the engine design may be optimized. The configurations analyzed are:

1) The baseline impeller rear cavity
2) The cavity with a baffle plate
3) The cavity with rearranged in and out flows.
3.1 The Baseline Impeller Rear Cavity

The baseline impeller rear cavity configuration is shown in Fig.1. Boundary a-b is the impeller rear surface where rotating wall boundary condition is applied. The velocity at this boundary is set to be equal to the surface velocity of the rotating wall. Boundary c-d is the combustor casing and boundary b-e is the bearing housing. Stationary turbulent wall boundary condition is applied to c-d and b-e, where no-slip condition is imposed. Flow enters the cavity at the impeller tip clearance a-c where the inlet mass flow and flow direction are specified. At the inlet the ratio of air tangential velocity to local disk tangential velocity, the so called K-factor, is equal to 0.8. Flow exits the cavity at d-e to turbine disks, where the exit static pressure is specified. Periodic boundary condition is applied to the front and the back meridian plane. For this case the the rotational Reynolds number $R_e$ is $1.6 \times 10^5$ and the mass flow parameter $C_a$ is $3.7 \times 10^4$.

The velocity vectors in the meridian plane are calculated. Based on these velocity vectors approximate streamlines are constructed with a post processor. Starting from one point, the streamline obtains a length increment along the local velocity vector direction, and then it obtains another length increment along the next nearest local velocity vector direction, and so on. With the present limited grid resolution the streamlines constructed in such a way are approximate streamlines. If the grid resolution increases, they will approach to the accurate streamlines. Although approximate, such streamlines illustrate the major flow features as is shown in Fig.1. Due to high tangential swirl there is a significant centrifugal force. In the middle of the cavity this centrifugal force is almost balanced by the radial pressure gradient, $\rho u^2/r = \partial \rho/\partial r$. However such balance is lost close to the solid wall. As is known, close to the wall the static pressure is constant across the boundary layer and is equal to the pressure of the local external flow. Therefore close to the wall the radial pressure gradient is maintained the same as in the middle of the cavity. At the rotating wall surface, the tangential velocity of the air is the same as the wall velocity which is faster than the tangential velocity of the air in the middle of the cavity. Hence the centrifugal force is larger than the pressure gradient, i.e. $\rho u^2/r > \partial \rho/\partial r$, so that the flow along the rotating wall is pumped outwards. On the contrary, at the stationary walls the air velocity is zero. Therefore the centrifugal force is less than the pressure gradient, i.e. $\rho u^2/r < \partial \rho/\partial r$, so that along the stationary wall the flow is pushed inwards. This generates the so-called secondary flow as is clearly shown in Fig.2 which is a magnification of the geometry close to the upper tip. Such secondary flow creates the complicated vortex patterns of Fig.1, which in turn generates substantial mixing. A portion of the inlet flow stays close to the stationary upper wall and reaches the exit without much mixing. The rest of the flow turns radially inward and back to the middle of the cavity where secondary flow mixing takes place. There is intensive mixing at the top of the cavity as is indicated by the strong vortex shown in Fig.3, while there is relatively less mixing activity at the bottom of the cavity. Fig.1 also shows that at the bottom of the cavity the flow is recirculating with very little through flow. This could result in a high total temperature region at the bottom due to the windage (i.e. the friction work), because the wall friction is doing work on the flow there but very little energy is convected away from that region. Such a high temperature zone may have a negative impact on impeller durability.
The static pressure contours are plotted in Fig.4. It is seen that all the contour lines are horizontal and normal to the radial direction. This shows that there is very little axial pressure gradient, and the cavity is dominated by the radial pressure gradient because of the high swirl. The static pressure increases with increasing radius as is shown in Fig.5. For this baseline configuration the static pressure in the cavity was measured in an engine test in the authors' company as is shown by the symbols on Fig.5. Considering that the measurements has certain error band, the calculated pressure agrees reasonably well with the measurement.

The important parameter describing the tangential swirl is the K-factor. The solid line in Fig.6 shows K-factor distribution with radius along a line through the middle of the cavity. As is seen, at the lower part of the cavity K-factor decreases with increasing \( r \) and the flow behaves like a free vortex. However at the upper part of the cavity K-factor increases with \( r \). At \( r/r_{tip} \) of 0.75 the K-factor has a minimum value of about 0.4. K-factor is directly related to the relative tangential velocity \( w_6 \). The dashed line in Fig.6 shows the non-dimensionized relative tangential velocity

\[
\frac{w_{6,air}}{w_{6,rev,tip}} = (1 - K)\frac{r\omega}{r_{tip}\omega}
\]

As is seen the relative tangential velocity is higher at the upper cavity.

### 3.2 The Impeller Rear Cavity with a Baffle Plate

The wall friction increases with the relative tangential velocity \( w_6 \) which is higher at the upper cavity according to Fig.6. The windage is the product of wall friction and wall velocity (\( r\omega \)). Therefore, most of the windage is generated in the upper cavity, say above \( r/r_{tip} = 0.75 \), which is in line with the result of disk friction theory. In order to reduce the windage which in turn reduces the cavity temperature, it is natural to concentrate in the upper cavity first. According to past experience, windage can be reduced by introducing a stationary baffle plate mounted close to the rotating surface. This produces a very narrow passage through which both the upward flow on the rotating surface and downward flow on the stationary surface pass. A baffle plate is positioned in the upper part of the impeller rear cavity as shown in Fig.7. This baffle plate is a circular ring which extends from \( r/r_{tip} \approx 0.75 \) to the top of the cavity. The distance from the baffle to the impeller rear surface is 6% of the radial length of the baffle plate. In order to understand how this baffle plate functions, the flow in the impeller rear cavity with such a baffle is calculated. The rotating Reynolds number, mass flow parameter and boundary conditions are the same as in the baseline configuration of section 3.1. The baffle plate as an internal object was blocked off in the calculation as is shown on the grid in Fig.7.
The approximate streamlines in meridian plane are shown in Fig.7. The baffle plate energizes the flow in the cavity, so that there is more secondary flow mixing compared with the baseline configuration of Fig.1. The strong vortex at the bottom in Fig.7 indicates significant increased mixing over the baseline. Therefore the low through flow problem described in section 3.1 disappears. Behind the baffle between the two stationary walls, there is a strong recirculation zone as is shown in Fig.8.

The static pressure contours plotted in Fig.9 show similarity to the baseline configuration. The cavity is dominated by radial pressure gradient with very little axial pressure variation. The static pressure distribution along the impeller rear surface is shown in Fig.10 and compared with the baseline configuration. In the upper cavity corresponding to the baffle region the radial pressure gradient is higher with the baffle than without the baffle. In this calculation the same exit pressure as in the baseline was used and therefore the resulting inlet pressure is higher. However, in reality the inlet pressure is the same as in the baseline, and hence the baffle results in lower pressure in the cavity and lower exit pressure. This reduces the axial load on the impeller, which in turn reduces the load on the thrust bearing holding the rotating assembly. The thrust bearing design and durability are affected by such load.

In Fig.11, the K-factor distribution is compared with and without baffle. As is seen in the lower cavity the K-factor are similar for both configurations. Although with baffle Fig.7 shows more mixing activity due to strong secondary vortex in this region, this vortex is in meridian plane and does not influence much the tangential component of velocity. However, in the upper cavity...
where the baffle is located, the K-factor is higher with the baffle than without the baffle. The high K-factor implies low relative tangential velocity which reduces the friction on the rotating surface. The windage is therefore reduced significantly with the baffle, which confirms the past experience with baffle plates. In other words, the baffle plate reduces the air total temperature in the cavity which has significant benefit on the impeller durability. The high tangential swirl associated with the high K-factor creates large radial pressure gradient. This is why with the baffle the radial pressure gradient is higher in the upper cavity corresponding to the baffle but is similar in the lower cavity compared with the no baffle situation, as shown earlier in Fig.10.

3.3 The Rearranged in and out Flow Configuration

The flow leaving the cavity at the right, Fig.1, is used for turbine cooling. Therefore, the total temperature of this flow is an important design parameter. As discussed earlier, most of the windage is generated in the upper part of the cavity. In the configuration described both in section 3.1 and 3.2, the windage temperature rise is added to the flow before it feeds the turbine disks region. In order to reduce the temperature of the flow to the turbine disks, a rearranged in and out flow configuration is proposed as is shown in Fig.12. In this configuration non-swirling inflow is introduced from the upper stationary wall with a specified mass flow. Another concern is that the impeller durability is limited by stress levels, especially near the bottom or hub. The hub temperature will be reduced, if the flow exits from the impeller tip instead of entering, because in this case, the windage temperature rise will accumulate from hub upwards rather than from tip downwards. Further, since windage work is low at hub, the air temperature rise there is also very low. Therefore the flow direction at the impeller tip is reversed in this proposed configuration and 44% of inlet flow exits at the impeller tip, while 38% of the inlet flow leaves from the right for turbine disk cooling. The rest of the inlet flow exits at the bottom where the exit static pressure is a boundary condition.

For this case the rotational Reynolds number $R_e$ is $1.6 \times 10^7$ and the mass flow parameter $C_{0}$ is $6.8 \times 10^4$.

The approximate streamlines in meridian plane is shown in Fig.12. One of the major characteristics is that the flow leaving at the right for turbine cooling does not mix with the flow on the left of the inlet so that no windage is added to it. Therefore, this flow is cooler compared to that in section 3.1 and 3.2. This fulfills the objective of this change in inlet flow location. Since flow exits at the impeller tip, the bulk flow direction close to the impeller rear surface is upwards, so that the windage is accumulating from hub upwards. Hence the hub is cooler and the tip is hotter compared with the baseline configuration. In the middle of the cavity, substantial mixing takes place as indicated by the strong vortices. Detailed review of the streamlines and vectors show rather interesting demarcation of flow. Fig.13 is a vector plot of upper part of the cavity. Two flow regions are clearly evident: upper clockwise and lower anticlockwise separated by a demarcation line. Such demarcation indicates that the bottom of the cavity will not be affected by the windage at the top of the cavity and remains cool although strong secondary flows exist. This fulfills the objective of reversing the flow at the impeller tip. By such a reversing, as a matter of fact, the air total temperature in the cavity is redistributed so that it is cooler in the region where low temperature is more demanded.

It is interesting to note that according to the simple disk friction theory, the impeller rear surface pumping flow equals 44% of the inlet flow for this case. If the flow exiting at the tip equals the pumping flow, no flow recirculation would be expected. However Fig.13 shows strong recirculation in the upper cavity. This indicates the inaccuracy of the simple disk friction theory in which a substantial inviscid core region is assumed.

The K-factor distribution is compared with the baseline configuration in Fig.14. Because the inlet flow is non-swirling, the K-factor throughout the cavity is much lower than that in the baseline configuration. This could result in large windage. However the temperature rise due to the windage can be controlled.

---

**Fig.11** K-factor distribution in the cavity

**Fig.12** Approximate streamlines in meridian plane
The rearranged in and out flows, Config.3
Fig. 13 The two vortices mix very little with each other by the flow amount exiting at the impeller tip. Increasing this flow reduces the temperature rise. It should be noted that the K-factor for this configuration increases monotonically from bottom to top.

The static pressure contours of Fig. 15, in contrast to Fig. 4 and Fig. 9, show that there is no dominant radial pressure gradient because of the low K-factor in the cavity. In fact the pressure gradient is very small in all directions and the static pressure in the whole cavity is almost constant. This results in high axial load on the impeller. However, considering the pressure variation in the baseline configuration, it is clear that the impeller axial load can be controlled by appropriate pre-swirl at the inlet. The higher pre-swirl at the inlet will result in higher radial pressure gradient, which in turn will reduce the axial load on the impeller. The pre-swirl will also influence K-factor and, hence, the windage heating.
4. CONCLUSIONS

The rear cavity of a centrifugal compressor impeller in an aero-engine provides quite a challenge to the aero-mechanical design. The flow is viscous driven and dominated by strong secondary flow. The flow behaviour has significant influence on the durability of impeller, turbine and thrust bearing. A 3-D Navier-Stokes CFD code has been applied to analyze the complicated flow in the impeller rear cavity of an aeroengine. The calculation agrees with available measurement data reasonably well. Three different configurations are analyzed. Details of the flow are obtained for each case in terms of secondary flow, vortex mixing, pressure distribution, K-factor distribution and windage. It is shown that the baseline configuration results in a hot region near the hub. A stationary baffle plate located near the impeller tip reduces windage and, at the same time, reduces axial load on the impeller. By introducing the inflow from the upper stationary wall, the total temperature of air for turbine cooling can be reduced. By reversing the flow at the impeller tip, the temperature at the cavity bottom or hub can be reduced. The cavity tip temperature can be controlled by varying the flow exiting at the tip, while the axial load on the impeller can be controlled by inlet pre-swirl. The above results provide valuable information for optimizing the engine design.

ACKNOWLEDGMENT

The authors would like to thank ASC Inc. and especially Mr. P.F. Galpin for their excellent support. Permission to publish this paper by Pratt & Whitney Canada Inc. is gratefully appreciated.

REFERENCES